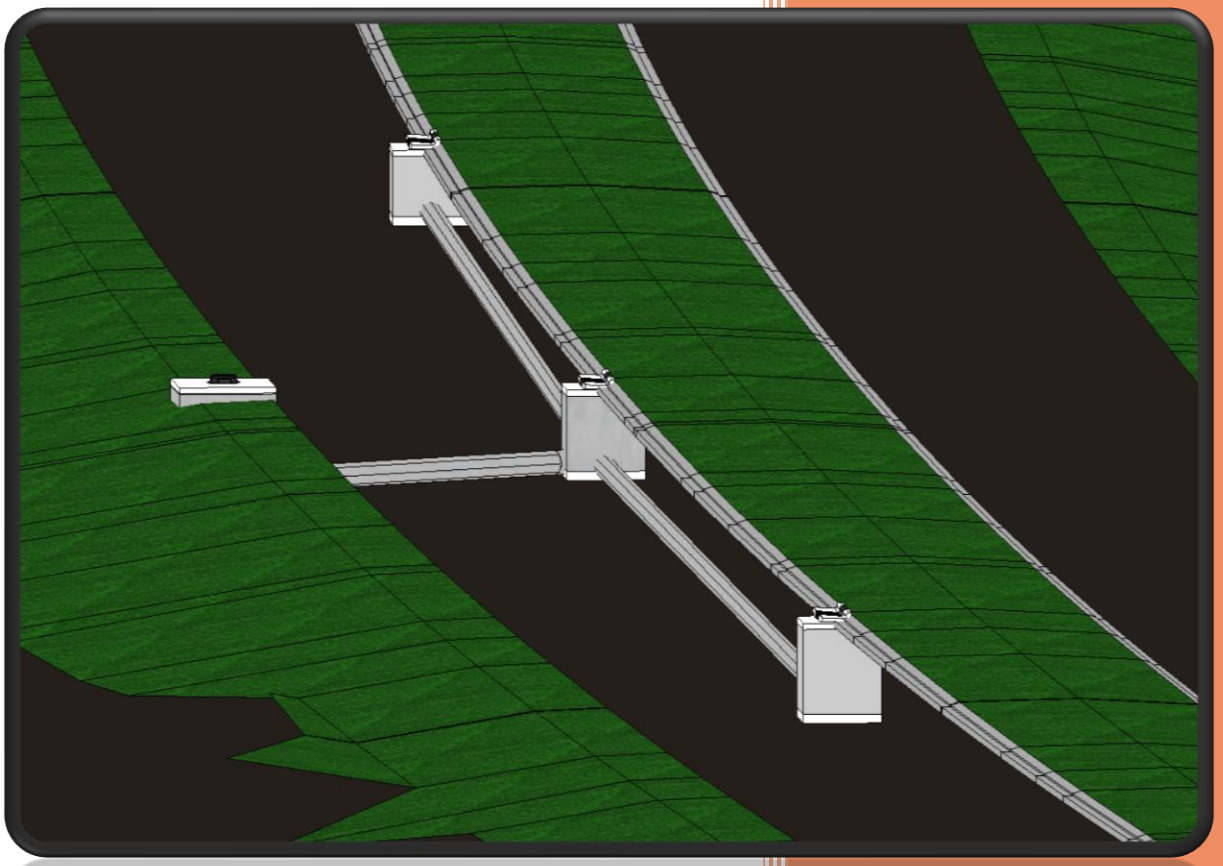
NCDOT

OpenRoads Drainage & Utilities Quick Start Guide



Bentley Sustaining Infrastructure

November 2021

STATE OF NORTH CAROLINA DEPARTMENT OF TRANSPORTATION

Table of Contents

Note: This document is meant to be a living document that will be updated as Bentley makes improvements and NCDOT projects are fully transitioned to ORD software. Screenshots and other new features may not be updated immediately with new software updates.

1.1 Introduction: OpenRoads Drainage and Utilities

Introduction and Differences Between Other Software

The Drainage and Utilities package in OpenRoads Designer (ORD) will take the place of Geopak Drainage for the design and analysis of certain hydrologic and hydraulic components for NCDOT projects. A primary difference from other software is that all drainage data, components, computations, etc. are stored within the .dgn file and not in a separate file (no .gdf or other files).

The following guide shows a new user the step-by-step process for inputting and analyzing the hydrologic and hydraulic components. Throughout this guide, there will be links to videos demonstrating different process and workflows. **It is assumed the designer is familiar with the hydraulic and hydrologic analysis/design and it is the designer's responsibility to adhere to all NCDOT Drainage Manual Standards.**

1.2 Introduction: Drainage and Utilities Workflow

Recommended Process for Designing and Analyzing within Drainage and Utilities

The following is the standard process recommended by NCDOT when completing the hydrologic and hydraulic design in ORD. This process contains the same methodologies and concepts used in all analysis software, but the sequence of inputs most closely resemble the workflow in StormCAD.

1. Create necessary terrain models
2. Place nodes
3. Add catchments
4. Place gutters to direct bypass
5. Run 4 in/hr scenario
6. Check spread and adjust inlets as necessary
7. Connect nodes with appropriate conduits
8. Check design constraints
9. Run design scenarios
10. Check drainage profiles
11. Adjust and run analysis scenarios
12. Produce flex tables for review

1.3 Introduction: Opening the Hydraulics Workspace

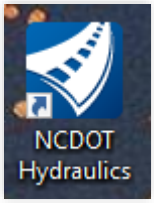
Getting Started in ORD with the Hydraulics Workspace

When beginning the workflow, the user needs to ensure the latest NCDOT workspace is installed on the computer they will be using. The latest NCDOT ORD workspace and installation instructions can be found here:

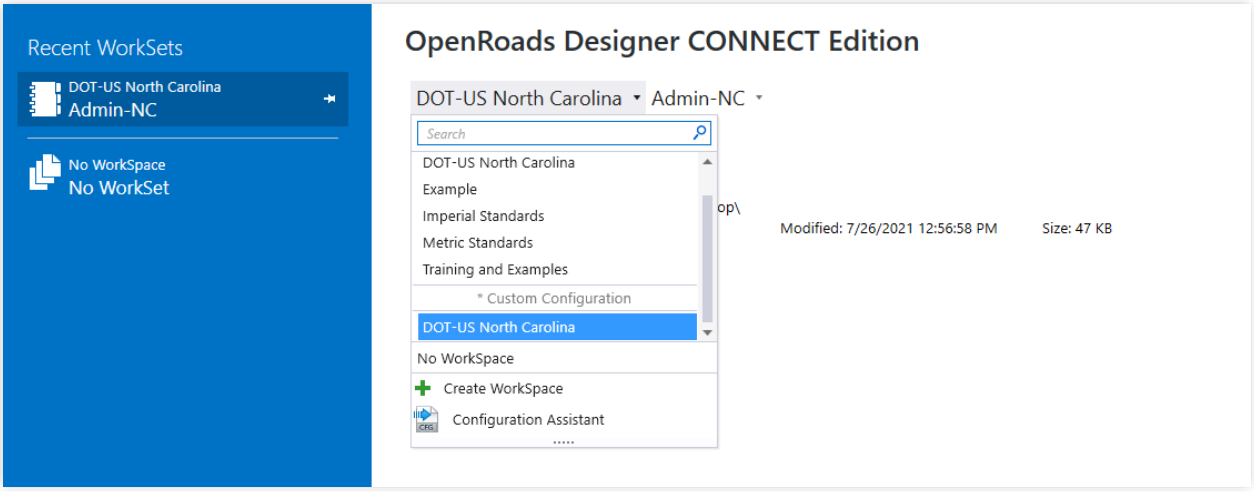
- https://connect.ncdot.gov/resources/CADD/Pages/default.aspx

Once ORD has been properly installed on the user's computer and the guidance followed from the link above, the user should end up with an "NCDOT Hydraulics Shortcut on their desktop as shown below.

- Double click the NCDOT Hydraulics ORD Shortcut



- Select the DOT-US North Carolina Workspace



- Select the appropriate WorkSet based on the project the user is working on
- Create new file as appropriate (**Section 2.1**, **Section 3.1**)
- **Projectwise integration and managed workspaces are still under development. Sections will be added to future versions of this document with additional Projectwise direction**

2.1 Terrains: Creating Terrain Models From Survey .Tin Files

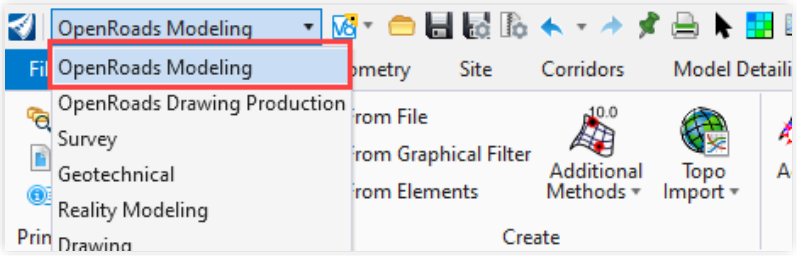
Creating Contours and 3D Terrains from .tin Files

The following is a brief overview of how to create terrains from a typical survey file in the .tin file format. A terrain in ORD will be used similarly to how a contour file was used in Microstation V8i but, also contains the capabilities of a 3D surface.

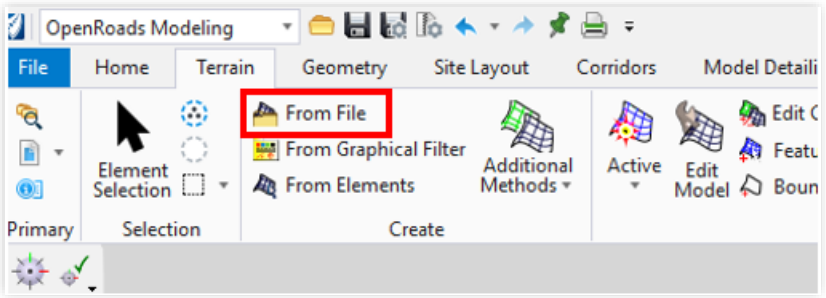
- Create a new .dgn file named *project #_survey_terrain* from the NCDOT 3D seed file

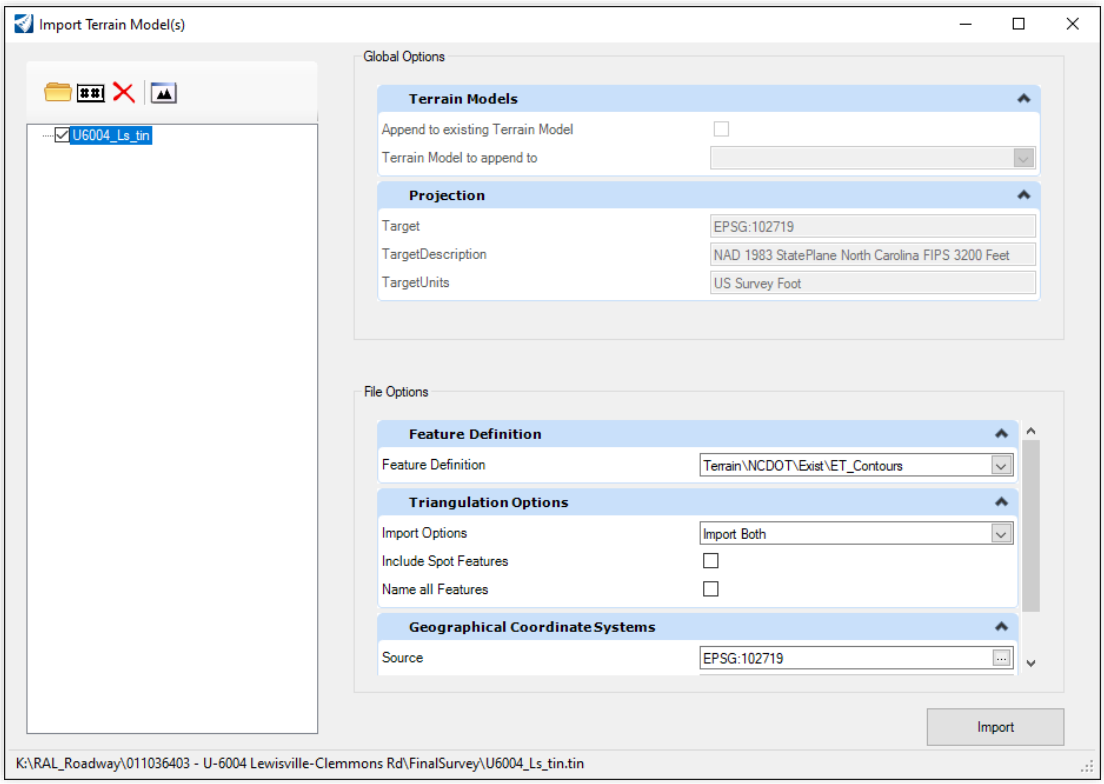
Helpful Hint: When working with Terrains, do not create/generate multiple terrains in a single file. Only having one terrain in an individual .dgn file will help speed up processing and design.

- Select the OpenRoads Modeling Workflow from ORD Workflow drop down menu.



Navigate to Terrain Ribbon Tab > Create Tool Group > From File. Path to and select the appropriate .tin file




Import Terrain Model(s)
U6004_Ls_tin
Global Options
Terrain Models
Append to existing Terrain Model
Terrain Model to append to
Projection
Target
EPSG:102719
TargetDescription
NAD 1983 StatePlane North Carolina FIPS 3200 Feet
TargetUnits
US Survey Foot
File Options
Feature Definition
Feature Definition
Terrain\NCDOT\Exist\ET_Contours
Triangulation Options
Import Options
Import Both
Include Spot Features
Name all Features
Geographical Coordinate Systems
Source
EPSG:102719
Import
K:\RAL_Roadway\011036403 - U-6004 Lewisville-Clemmons Rd\FinalSurvey\U6004_Ls_tin.tin

Global/File Options	Typical User Selection
Filter>Source File Units	▪ *Most likely will be US Survey Feet for NCDOT Projects*
Feature Definition>Feature Definition	▪ *Select the "ET_Contours" for existing ground*
Triangulation Options>Import Options	▪ *Import Both*
Triangulation Options>Include Spot Features	▪ *Leave unchecked*
Triangulation Options>Name all Features	▪ *Leave unchecked*
Geographical Coordinate Systems>Source	▪ *Select North Carolina State Plane Coordinate System (Feet) EPSG:102719*

- Select Import and check to see that the terrain was imported successfully by visually inspecting the contours and by referencing in the survey or other project file to ensure proper location. Exit out of the Import Terrain Model dialog box.

Helpful Hint: If the user has a .dat file instead of a .tin file, follow the same steps as above but select the .dat file instead of the .tin file and for the "Edge Method" select "Remove Slivers"

2.2 Terrains: Creating Terrain Models From LiDAR

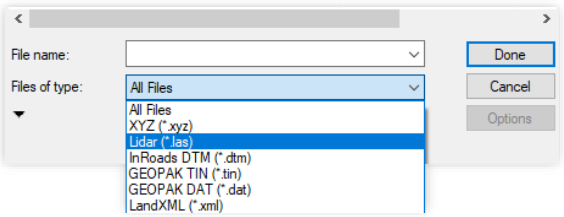
Creating Terrains from .las Files for Offsite Drainage and Contours

Drainage design often requires terrain data outside of the project survey limits. 2014 to 2017 QL1/QL2 LiDAR data is typically downloaded from North Carolina's Spatial Data Download Center as bare earth **.las** files. Legacy LiDAR files from 2001-2005 are available on the FRIS Data Download Center as a **.txt** file.

NCDOT Internal users may wish to continue to use the ArcMap Tool Boxes, "Get Decimated Lidar" and "Extract DEM", to create a .dat file. Users can create an Existing Terrain Model from a .dat file by following the helpful hint on the previous page.

- To create a terrain from a **.las** file, follow the first three steps of **Section 2.1** above but select the **.las** file instead of the **.tin** file

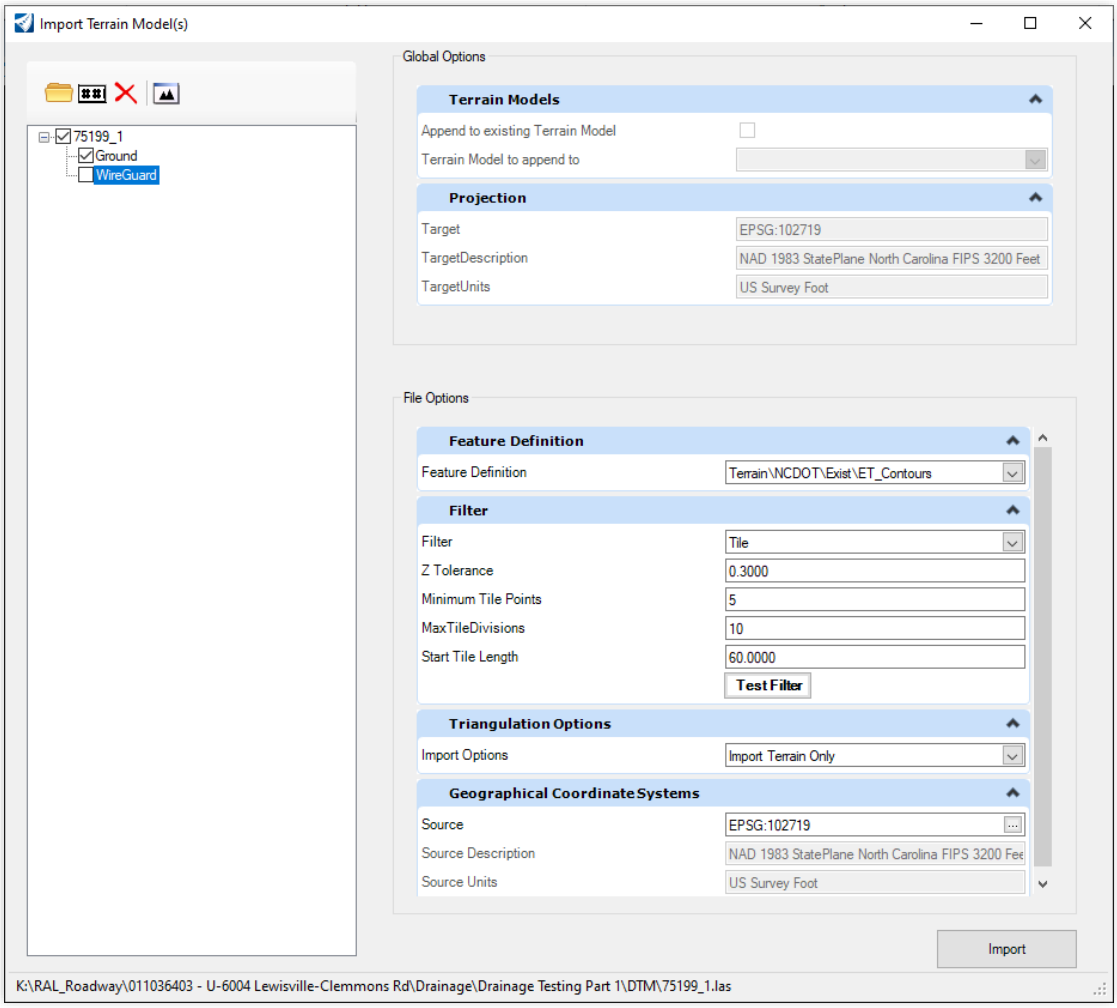
Helpful Hint: If the user cannot select a .las, .tin, or .dat file they may need to use the drop down in file explorer to change selection from "All files" to only ".las" (see screenshot below)



- Most QL1/Ql2 .las tiles downloaded from the NC Spatial Data Center are large (>100 MB per square mile) For large LiDAR Files, choose "import terrain only" (as shown in the screenshot below) under triangulation options. This will allow for filtering that will make data more manageable.

See Next Page for Screenshots

- Typical Global/File options are slightly different for a **.las** and are shown below.



- Filtering LiDAR will make working with the terrain model faster/easier and can still retain a high enough level of quality for most drainage applications.
    - The filter options shown above is for example purposes only. These settings reduce a typical NC LiDAR file by about 75%.
    - Different project types will require different levels of filtering.

Helpful Hints:

If downloading FRIS legacy LiDAR in the form of a **.txt** file, the file will have to be converted to a **.xyz** and can then be selected and imported using the same process as shown above.

The user can initially ignore the "source file units" field under the filter tab. It will disappear when the geographical coordinate system EPSG:102719 is selected.

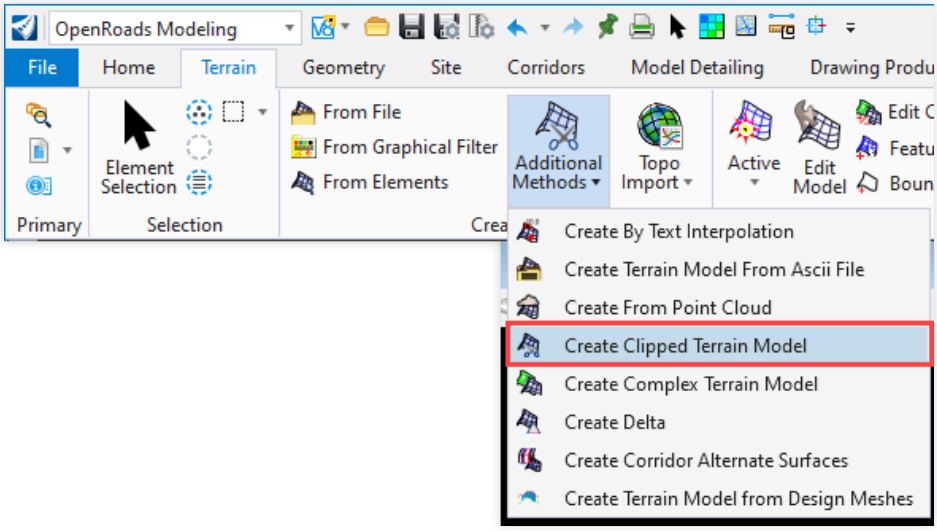
2.3 Terrains: Clip Terrain Models

Clipping Unnecessary Data from a Terrain

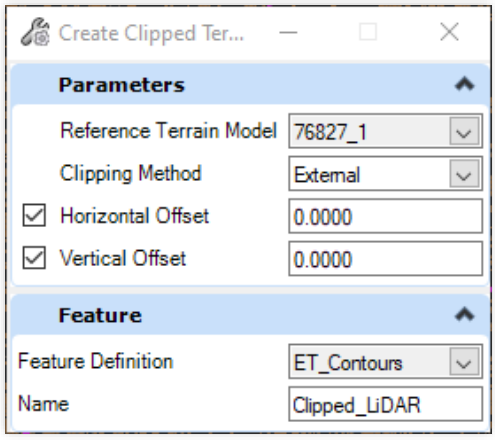
It is often the case that LiDAR tiles are much larger than necessary for the project limits. To reduce file size and memory usage, the user can clip a terrain by following the steps below:

Draw a shape of the area to be clipped (shape can be on any level).

- Select the OpenRoads Modeling Workflow from ORD Workflow drop down menu.
- Select the Terrain Ribbon Tab > Additional Methods > Create Clipped Terrain Model as pictured below.

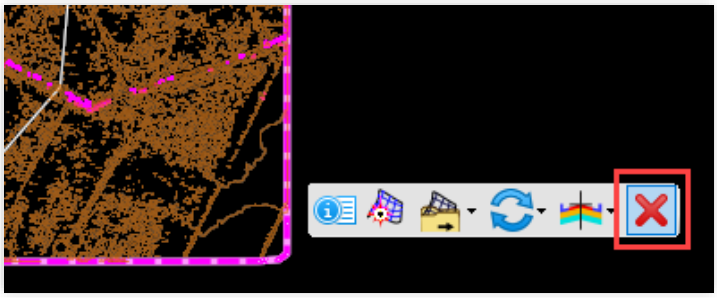


- The Create Clipped Terrain dialog box will open



Clip Terrain Option	Description
Reference Terrain Model	▪ *Select the terrain model to be clipped*
Clipping Method	▪ *Use "External" to delete everything outside of the shape drawn* ▪ *Use "Internal" to delete anything inside the shape drawn*
Horizontal and Vertical offset	▪ *Typically set to zero. If set to any other number, the terrain will be clipped by an offset of the shape drawn*
Feature Definition	▪ *For existing terrains use the ET_Contours feature definition*
Name	▪ *Name of the terrain i.e. "Clipped_Terrain_#"*

- Click through the options until the clipped terrain is created. It will be created on top of the original terrain so the user will need to delete the original (unclipped terrain) by selecting it, hovering over it and then clicking delete as shown below.



- In addition to this, delete the shape that was used to create the clipped terrain

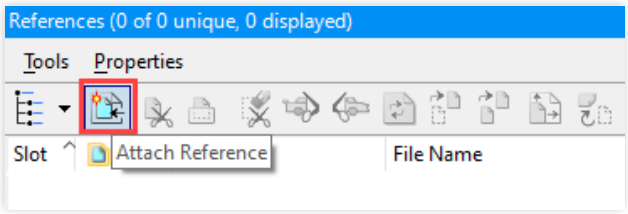
2.4 Terrains: Merge Terrain Models

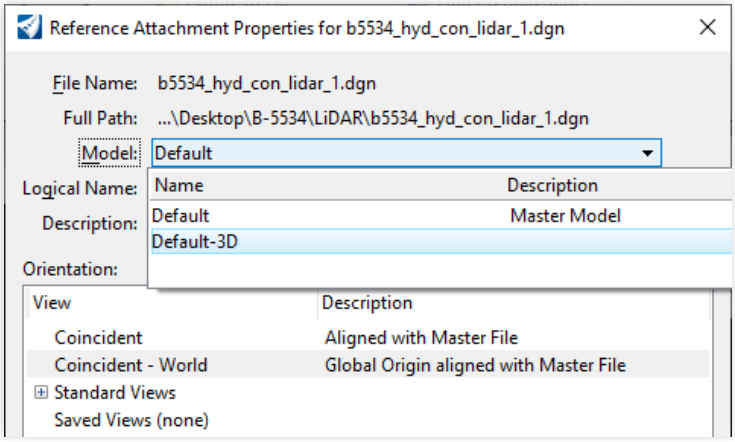
Combine Multiple Terrains into One

It is often helpful to merge terrains to create a model that has data from multiple terrains (LiDAR, Existing, Proposed, etc). At the beginning stage, the survey terrain is typically merged with the LiDAR. If the user does not need to merge terrains, this section can be skipped.

Before starting, the user must ensure that each terrain model to be merged is in a separate .dgn file. A new .dgn (3D seed file) should be created that will house the merged terrain model.

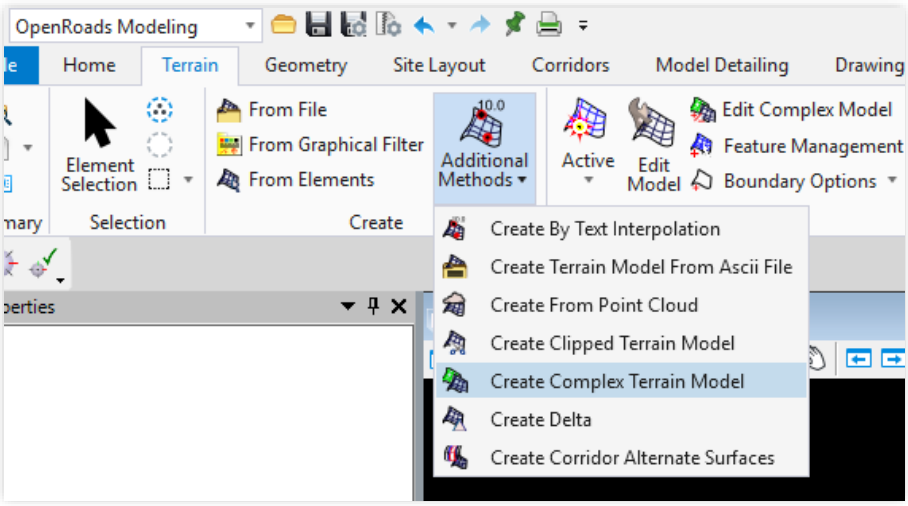
- To begin, open the new, blank .dgn file and reference in each terrain model that needs to be merged.



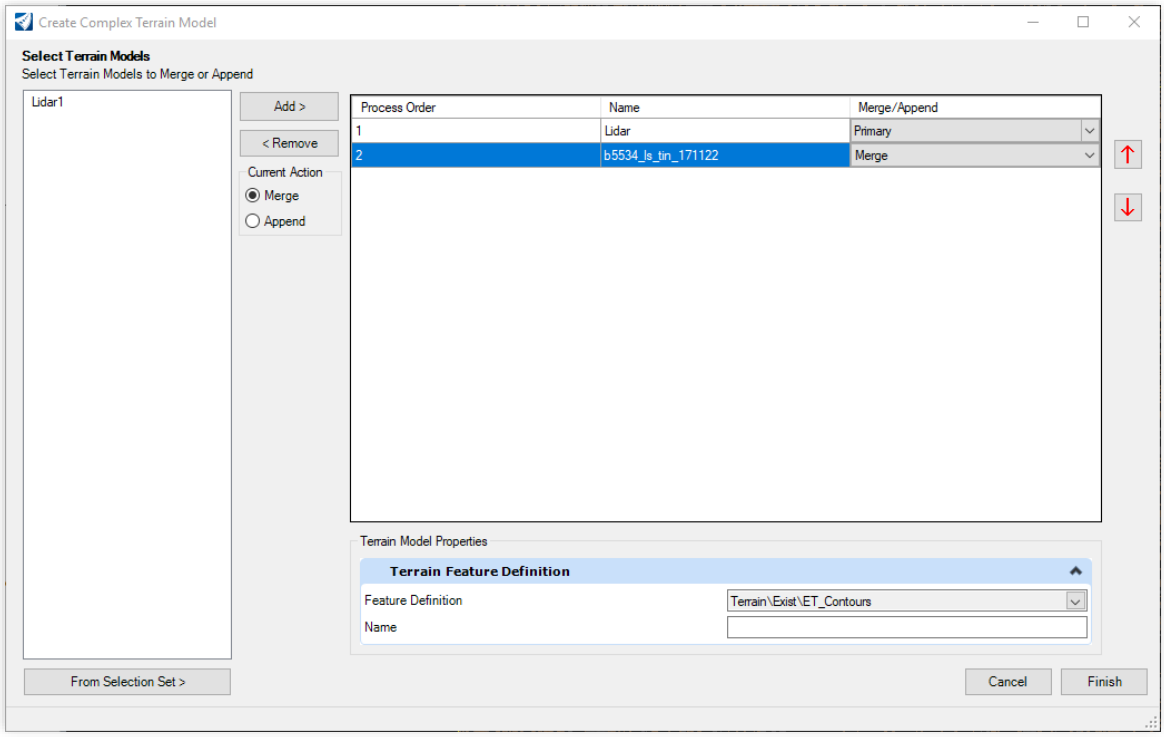


- Select the OpenRoads Modeling Workflow from ORD Workflow drop down menu.

- To Merge terrains, select Terrain Ribbon Tab > Create Tool Group > Additional Methods > Create Complex Terrain Model



- Add the "Primary" terrain model to begin. The Primary terrain model serves as the base terrain. Any data that overlaps the Primary terrain model will govern and over-write the Primary terrain model data. Add terrain model(s) to Merge and add to the process order.



- Select appropriate feature definition and select finish.

Helpful Hint: Merge action will overwrite the primary data where the models overlap. Append action should be used when user needs to add to the primary data (ex. Combining LiDAR panels that are next to each other but not overlapping).

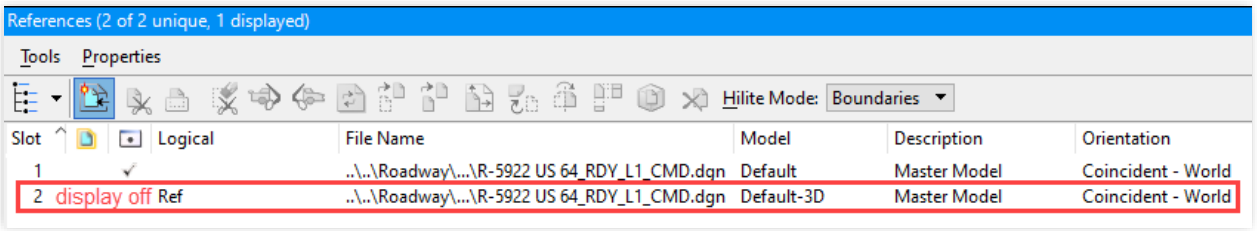
2.5 Terrains: Proposed Terrain Models

Creating Proposed Contours and Terrain Based on a Roadway Design Model

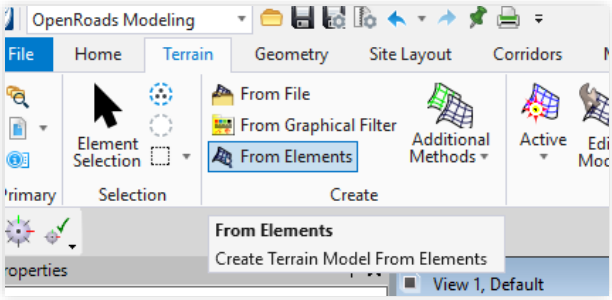
If a proposed terrain model is not provided for the user by the roadway designer, it can be created using the CMD (Corridor Model) file from roadway. A new .dgn (3D seed file) should be created that will house the proposed terrain model. It is often useful to create a proposed terrain and then merge it with the survey-LiDAR terrain as outlined in **Section 2.4** above.

- Reference the CMD file and turn off all levels that are not associated with the top surface (subgrade levels, grading levels, default, etc.).

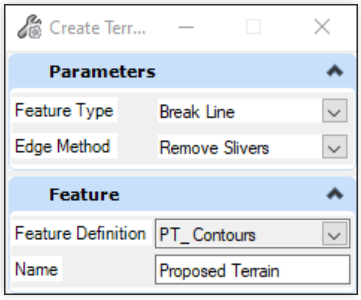
Helpful Hint: Turning off the 3D version of the roadway CMD Design file will turn off most subsurface roadway levels so that they are not accidentally selected (see screenshot below)



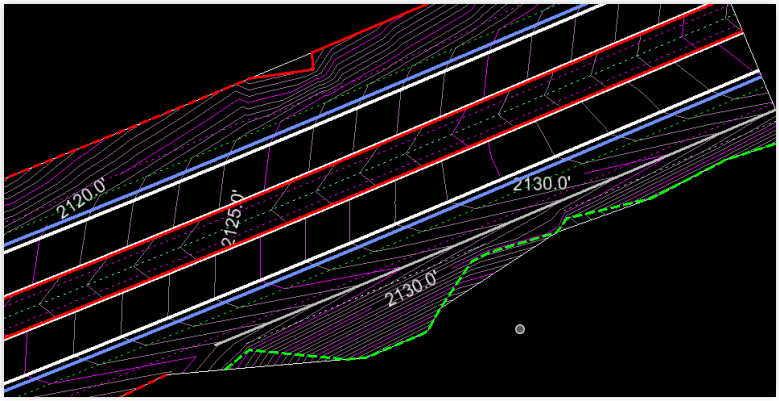
- Highlight all the elements (using the element selection tool) from the CMD file that were isolated in the first step and keep them highlighted for the next few steps.
- Select the OpenRoads Modeling Workflow from ORD Workflow drop down menu.
- To create the proposed terrain, select Terrain Ribbon Tab > Create Tool Group > From Elements



- Select feature type "break line", appropriate feature definition "PT_Contours" and other settings as shown below

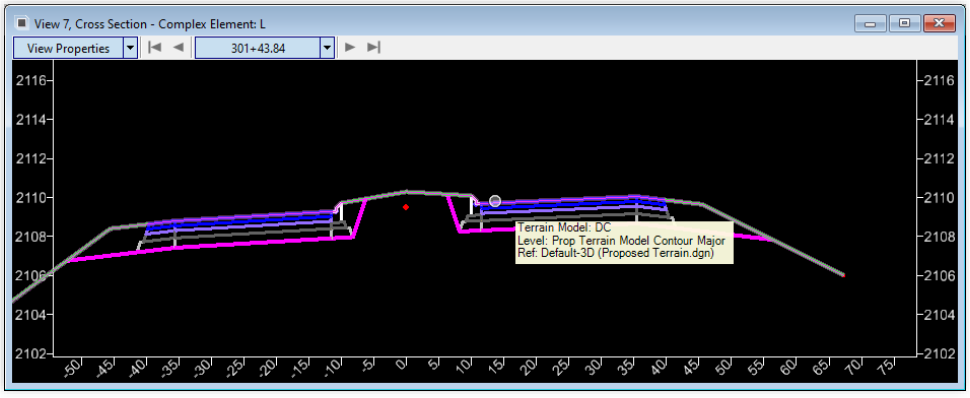


- Left click through the options until the proposed terrain is created



- Open the cross section view for the corridor model and compare it to the created terrain for accuracy (see video below)

  VIDEO LINK

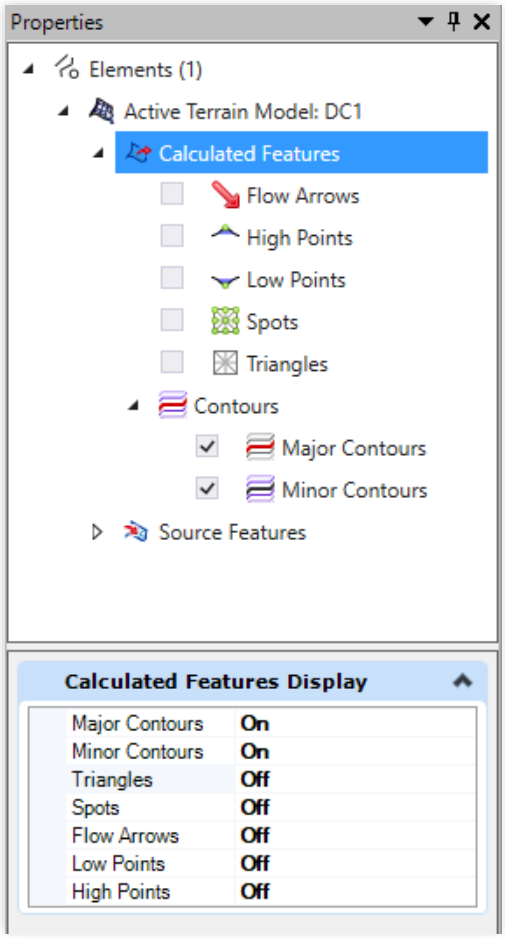


Note: The proposed terrain is live-linked / connected to the CMD elements that it is built from**. If roadway makes profile changes and then updates the corridor elements, the proposed terrain will automatically update with it**.

2.6 Terrains: Display Contours

Contour displays, as well as contour text, can be turned on/off as follows:

- Select the limits of the terrain model and display properties (for guidance on how to open properties see **Section 3.2**)

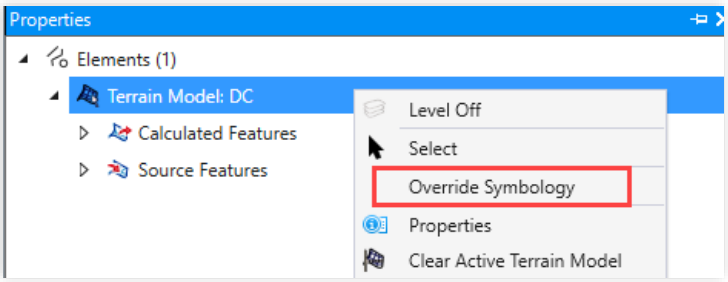


- Selecting Contours allows the user to display major and/or minor contours as well as turn on and off text.

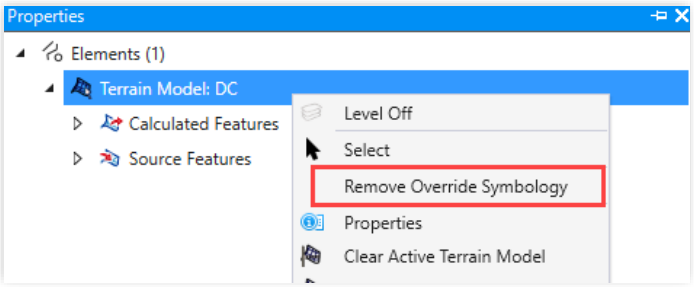
- Typically, the .dgn that houses the terrain of interest will be referenced into the drainage .dgn or another .dgn that the user is working in.

    - When this is the case, the user should follow the steps on the next page to change the contour settings

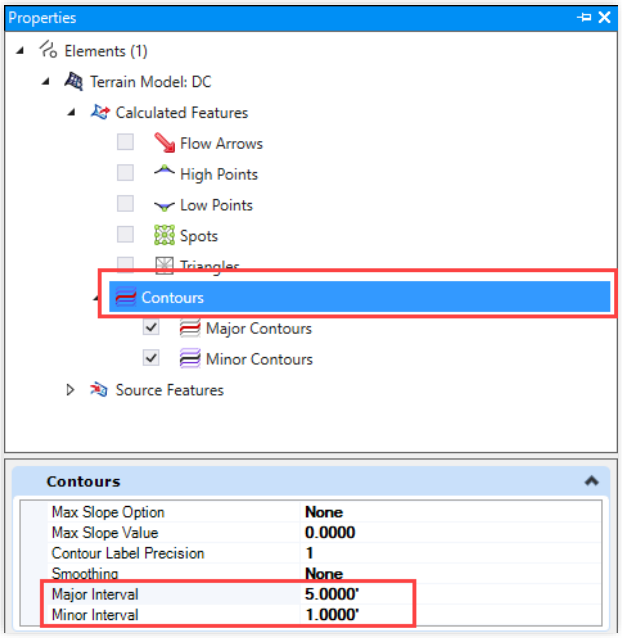
- Right click the terrain model in the properties window and select "Override Symbology" as shown below.



- The terrain display properties can now be customized for the drawing you are in. To revert back to settings in the actual terrain file, follow the same steps except select "Remove Override Symbology" as shown below



- To change the contours major and minor intervals, expand the calculated features tab and select just the "contour" tab as shown in the screen shot below



- The major and minor interval can be customized by user entering the values (default values are 5.0' and 1.0')

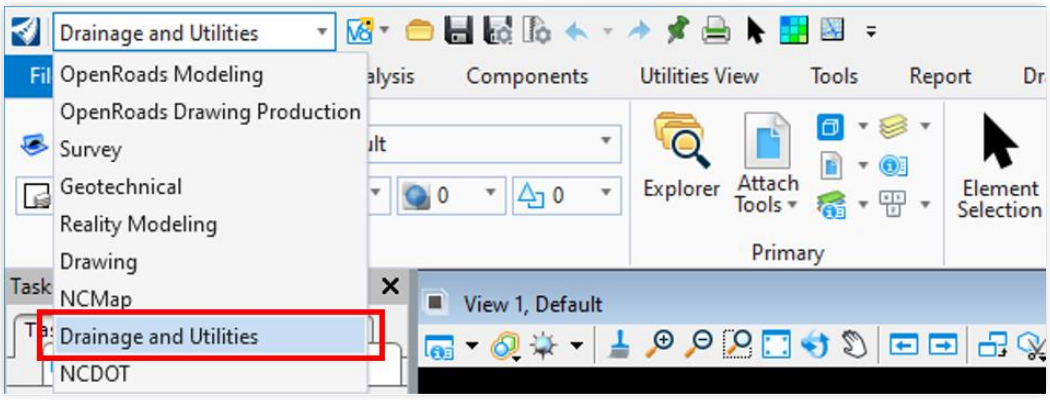
3.1 Drainage Modeling Initial Setup: Workflow

Setting the Active Workflow

The Drainage and Utilities software is embedded in each drawing file. If a DRN file has not been created for the project, create a new .dgn file named *project #_drn* from a NCDOT 2D seed file.

Helpful Hint: Since all drainage data is stored within the .dgn, always create a new .dgn file from an NCDOT 2D seed file when beginning a new project rather than copying in an old "go-by" project file.

To activate this software and create a new model, select the Drainage and Utilities Workflow from ORD Workflow drop down menu as shown below.



- Once Selected, the ribbon bar tabs at the top will change to the default Drainage and Utilities
- **If it is a new design file, the user will need to select a Drainage and Utilities component and click to insert it into the drawing (follow video instructions below). The user will be prompted to "Create Dainage and Utilities Project" that will reside within the drainage .dgn file. When working in the NCDOT workspace, creating a utility model will embed NCDOT drainage libraries and setting into the .dgn file.**

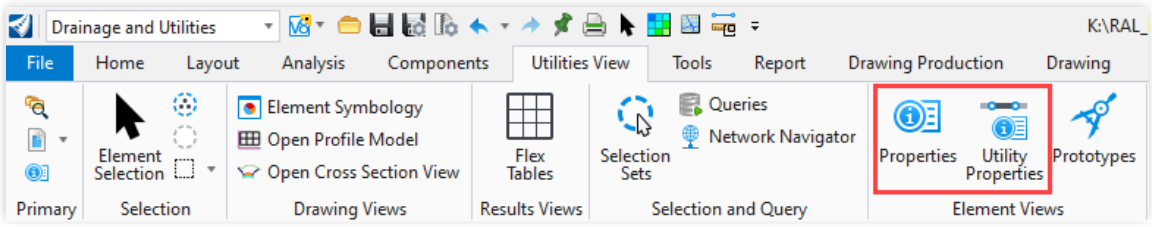
  VIDEO LINK

3.2 Drainage Modeling Initial Setup: Property Tools

Accessing and Setting Up the Commonly Used Property Tools

The Drainage and Utilities workspace utilizes two different types of property windows to edit all the drainage element properties (nodes, conduits, catchments, etc.) The steps below outline how to access the two most used windows: **properties,** and **utility properties**

- To access these property windows, go to the Utilities View Ribbon Tab and select both options outlined in red in the screenshot below.



Properties: The standard informational properties that most users are familiar with (CADD properties such as level, color, feature definition, etc.)

Utility Properties: Every available hydraulic property for a Drainage and Utilities element including customized properties. For example: an inlet would have variables such as spread criteria, elevations, efficiency, freeboard, and more which can be set using this window.

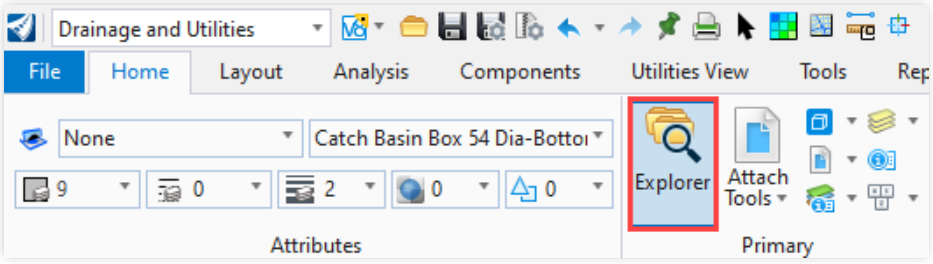
- Both property windows will be used **extensively** throughout the drainage design. It is useful for both windows to be docked within the ORD interface for quick access by simply clicking and dragging them to the sides or tops of the screen.

3.3 Drainage Modeling Initial Setup: Explorer Tool

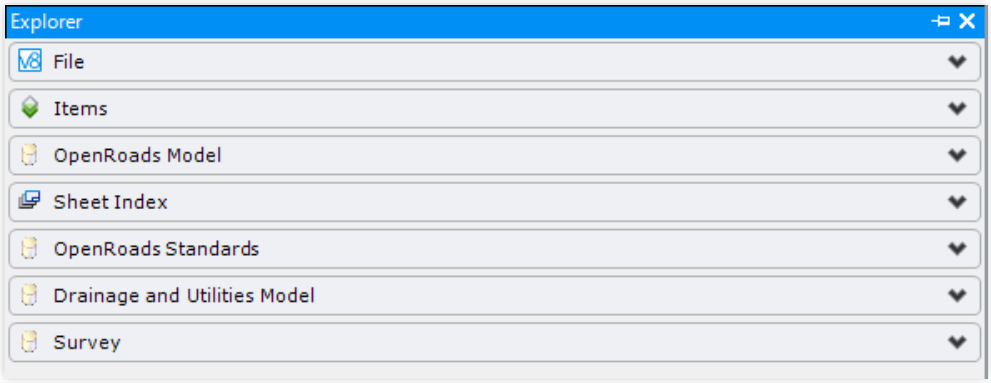
Accessing and Setting Up the Commonly Used Explorer Tool

The explorer tool is used to view all of the drainage components housed within the .dgn, many other components of the .dgn file and also components of any attached references.

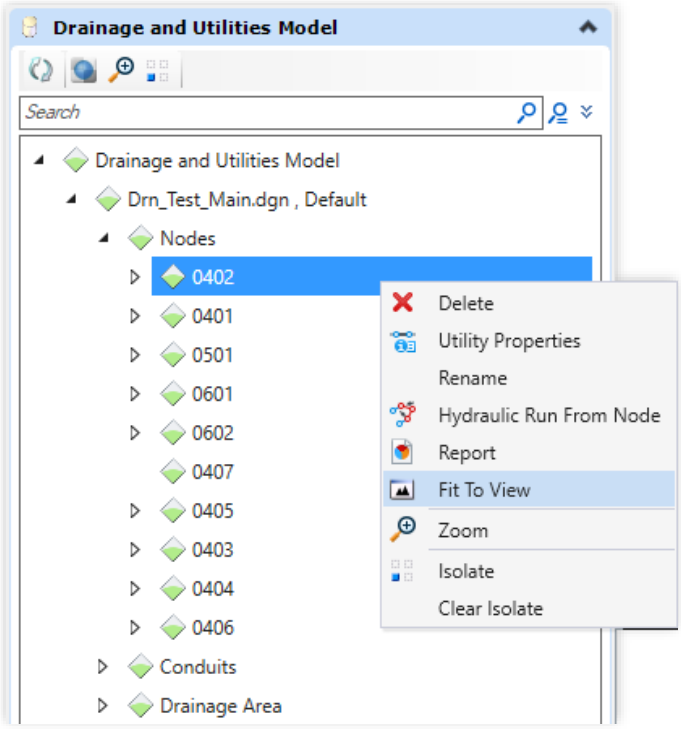
- To open the explorer tool navigate to the Home Ribbon tab > Explorer



- The explorer window will open.



- The explorer tool contains many different ways to navigate the components housed within the current .dgn. For example: Once a drainage system is created by following the rest of this guide, under the "Drainage and Utilities Model" tab, all the components are listed and can be right clicked, navigated to, etc.



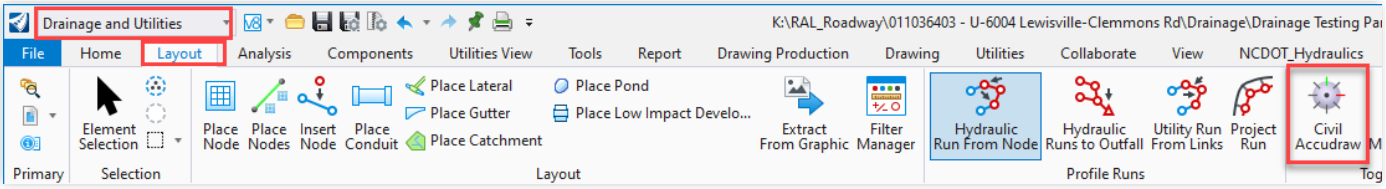
- The explorer window will be used frequently. It is useful to dock this tool within the ORD interface for easy access by simply clicking and dragging it to the sides or tops of the screen.

3.4 Drainage Modeling Initial Setup: Other Helpful Setups

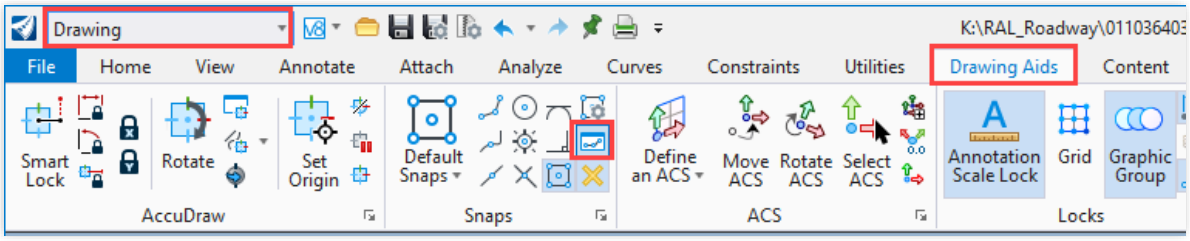
Opening Common Toolbars for the First Time User and Using 2 Screens

Other common toolbars that are used in drainage design and general drawing design are outlined below. It can be useful to dock these toolbars since they will be used extensively. In addition, opening two applications windows (helpful for dual screens, or even triple screens) is outlined below.

- Civil Accudraw will be used to place nodes and as a station and offset finding tool. The screenshot below shows one of several ways to locate the tool and activate the toolbar.

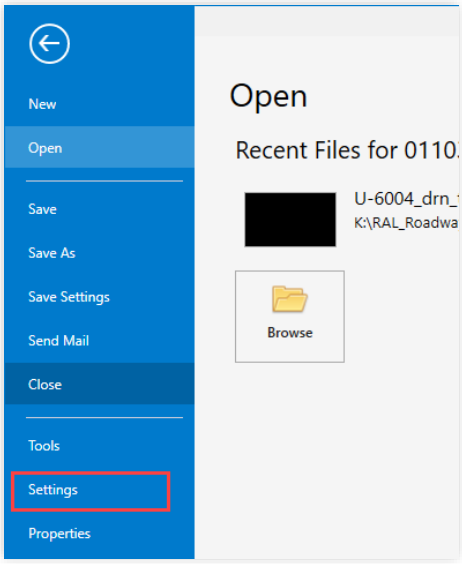


- AccuSnap will be used to lock on to key features and help place lines, nodes, and draw drainage areas. The screenshot below shows one of several ways to

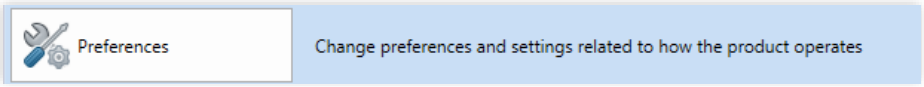


- OpenRoads has one application window by default the first time a user installs the software. To set up multiple application windows for the use of two or even three screens (helpful for cross section/profile alongside plan view), follow the steps below.

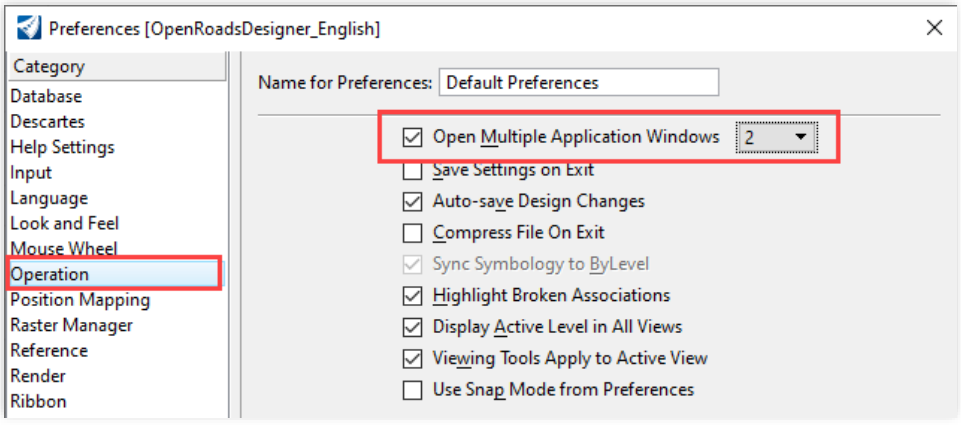
- Navigate to File > Settings as shown below.



- Select "Preferences" as shown below.



- Select "Operation" on the left panel and then check the box that states "Open Multiple Application Windows" as shown below.



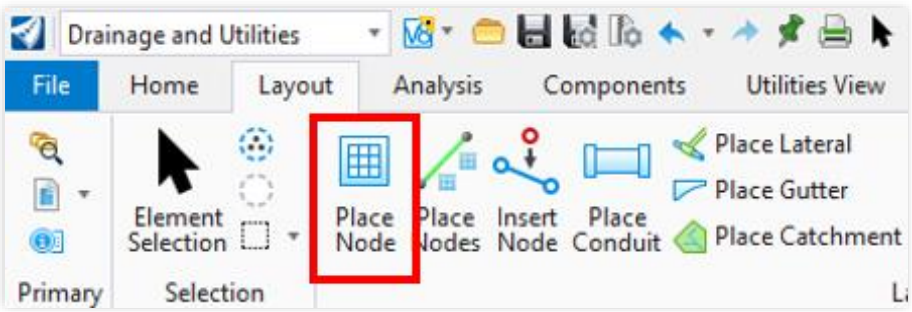
- Select how many application windows will open by default each time OpenRoads is launched as shown above.

4.1 Nodes: Placement

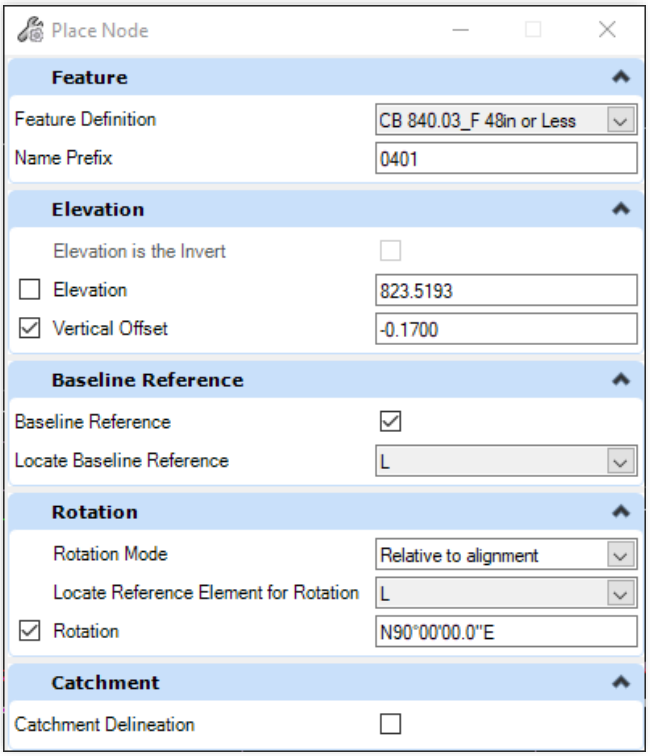
Placing Nodes to be Referenced to Alignment and Model Elements

In order for nodes to have top elevations calculated based on the proposed design and be referenced to the roadway alignment, they must be placed using the steps described below.

- Select the Drainage and Utilities Workflow from ORD Workflow drop down menu. (**Section 3.1**)
- To access nodes, go to the Layout Ribbon Tab > Place Node



- The Place Node dialog box will open. This contains initial settings for the node components of Drainage and Utilities.



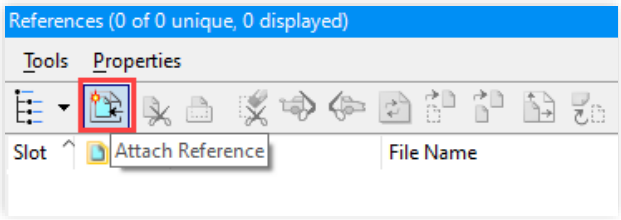
Place Node Option	Description
Feature Definition	▪ *This is where the user will select the type of node to be placed.*
Name Prefix	▪ *This is where the user will name/number the node. NCDOT typical structure numbering should be followed (for example this node is named 0401). Keep an eye out for future versions of this document should recommended naming convention change based on labeling standards.*
Elevation	▪ *Elevation can be set manually here; however, this will be left unchecked most of the time. Elevation of the node will typically be assigned based on a 3D linear element that is part of the roadway corridor model or the active terrain (see steps on next few pages for how to reference in terrains or roadway CMD file).*
Vertical Offset	▪ *If a constant elevation offset is needed such as gutter pan drop or local depression, that can be entered here. In the example above, a catch basin is set to -0.17' if the gutter flow line elevation reference will be used.*
Baseline Reference	▪ *Select the alignment from the drop-down list that the node's station and offset will be referenced from. Note: you must have the roadway alignment file referenced in for this to work (follow steps below).*
Rotation Mode	▪ *Choices are absolute or relative to alignment. Typically, relative to alignment will be used and the L or Y alignments can be selected in the drop down that appears.*
Rotation	▪ *Set the angle of rotation here.*
Catchment Delineation	▪ *This box can be checked to have the drainage area automatically created and auto delineated. This option only works when a terrain model will be selected for the node's elevation reference.* ***See the last paragraph of this section for more information on catchment auto-delineation.***

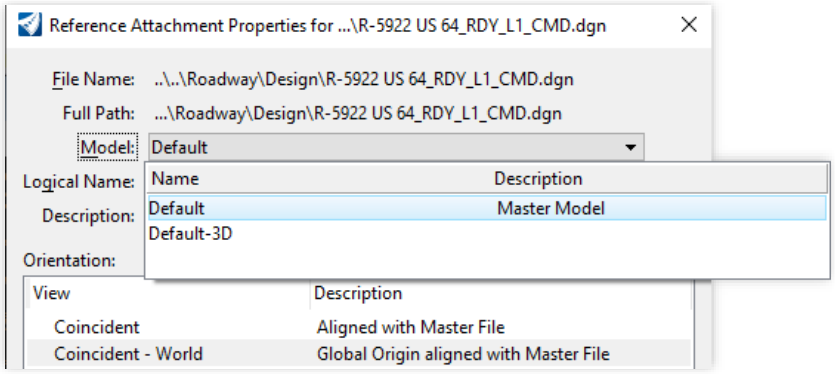
- Before placing a node, the user should activate Civil Accudraw as shown below. This will be used as a station/offset tool and will allow the location of the node to be input based on station and offset.

- The station-offset option is shown selected above however, it is recommended the user follow the steps in Appendix A – Customized Civil Accudraw to create a custom option that will improve the node placement experience for specific scenarios.

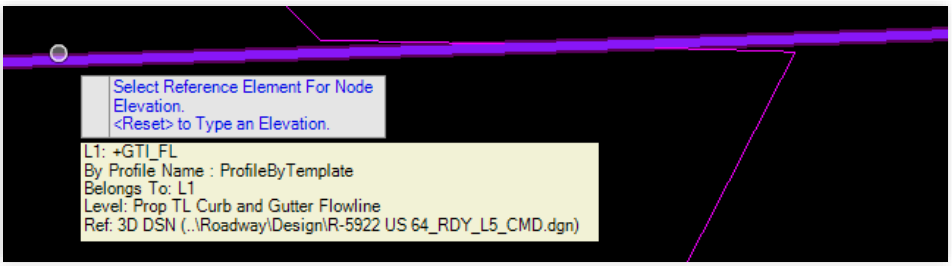
Helpful Hint: Ensure Civil Accudraw is associated with the current alignment first by using a command that will display it such as "draw line." If it is not associated with the current alignment, press the "O" key ("O" stands for origin) in either the station or offset data field, hit enter, and it will prompt the user to select an alignment. If entering "O" in the station field does not work, <reset> (right click) and try entering it in the offset field.

- Reference in the roadway CMD (corridor model) file (.dgn file) and/or the proposed/existing terrain model (.dgn file) using the attach reference tool.

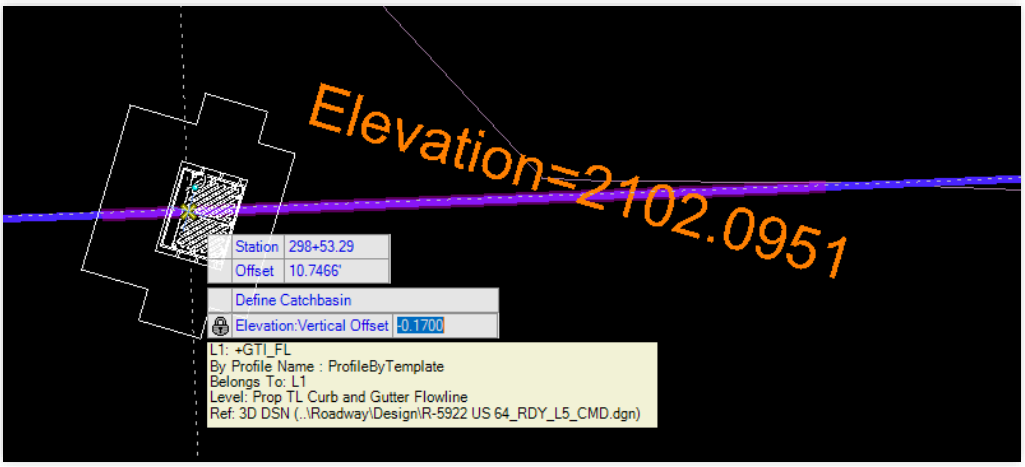




- "Default" can be selected. If the CMD .dgn file has a 3D model within it, it will automatically attach a 2D and 3D version of the reference.
- After the initial node options are set, the reference element for node elevation can be selected as shown below. Typically, one of three reference elements will be selected
    1. **A 3D linear element** *(In the picture below, a 3D linear element in the roadway CMD file that represents the gutter flow line is being selected as the reference.)*
    2. **A terrain model** (typically a merged combination of both proposed and existing terrains)
    3. **No reference** – user entered elevation (*shown as <reset> i.e. right click*)



Helpful Hint: If selecting a 3D linear element, use level display toggling and/or tentative Snap to ensure the appropriate element is being selected– it is very easy to select the wrong element – particularly in the curb line (the common level and element for catch basins to be referenced to will be the GTI_FL or another as shown in the screenshot

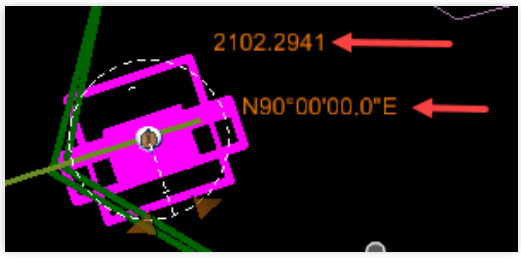


above) Select the elevation reference and vertical offset (-0.17' for catch basin's local depression), then enter the station and offset into the accudraw fields. Left-click to accept the parameters (location, vertical offset, rotation). Right-click through the steps at any time to exit the command.

- The placement of the node is now complete
- In addition to the steps above, a short video tutorial is available at the link below to provide further clarity on node placement.

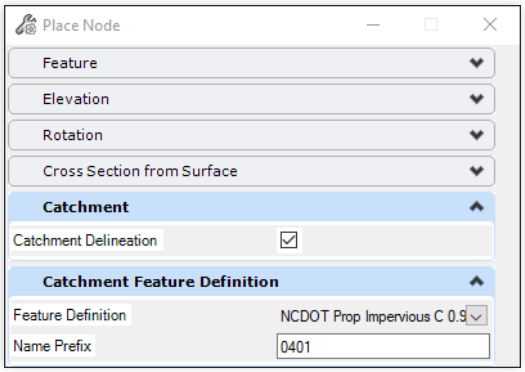
  VIDEO LINK

Helpful Hint: After a node has been placed, it can be selected, and basic elevation/ location properties will show up in orange text (as shown below). This feature can be utilized when needing to quickly adjust the rotation or elevation of a node. If the node has an elevation reference (3D element or terrain), the orange text elevation field will be a space to edit the offset elevation from the surface. If not, the field will simply be the ground elevation of the node. The node rotation arrow handles can also be used to rotate the node by clicking and dragging.



Auto delineation When Placing A Node Referenced to A Terrain

If the user is selecting a terrain model as the elevation reference instead of a 3D linear element, the catchment delineation feature can be used as shown below.

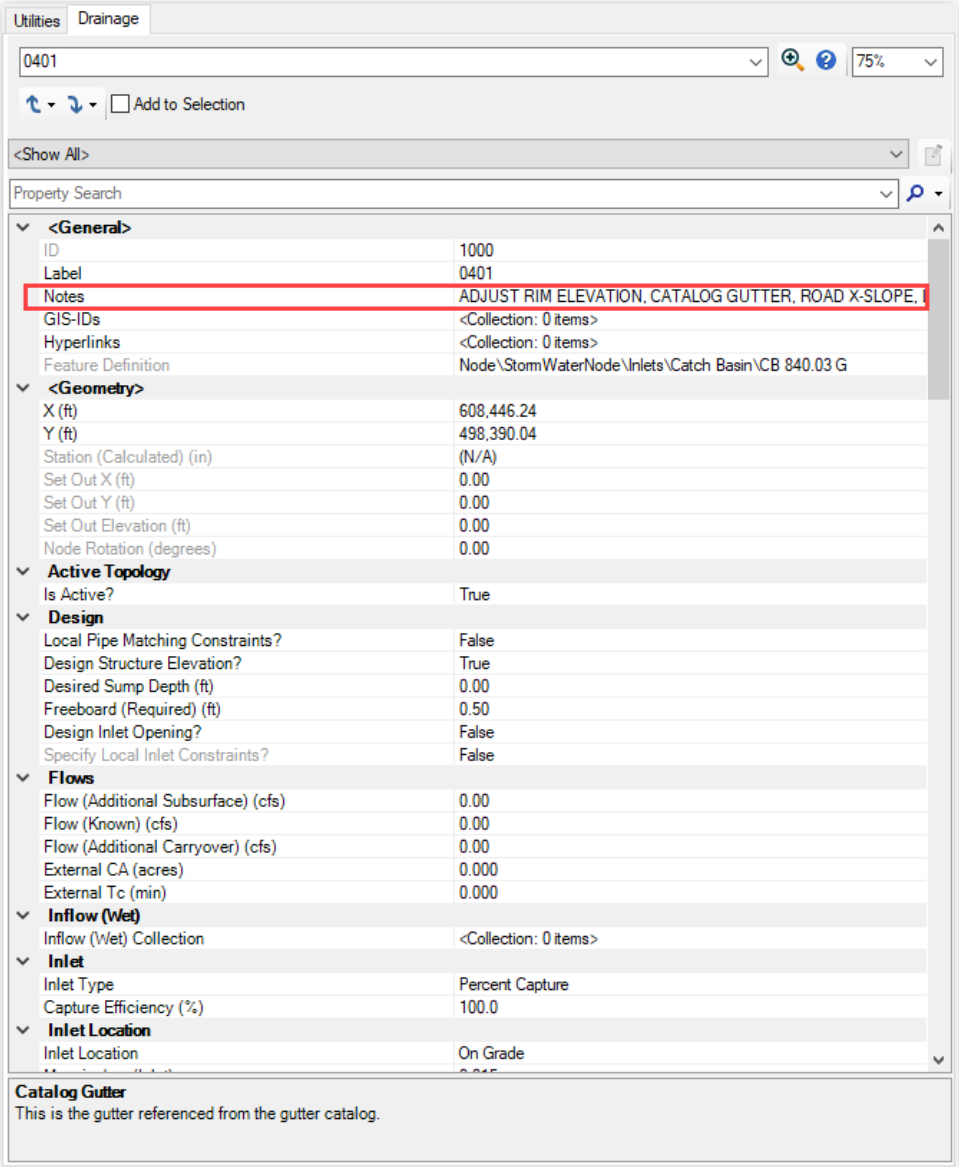


This tool can be powerful and save time in creating and delineating catchments (Section 5.1). **This tool should be used at the user's own risk and the drainage areas should still be checked thoroughly and tweaked as needed since they are being delineated by the software and not the engineer. At this time, there still appear to be some restrictions that are associated with this tool such as the inability to manipulate the catchment vertices after creation. This guide will be updated based on Bentley's future bug fixes and best practices. Please see Section 16.1 for NCDOT staff contacts to report bugs or other issues.**

4.2 Nodes: Node Hydraulics / Node Properties

Setting Spread Criteria, Rim Elevations, Freeboard Etc.

Select a drainage node by clicking on it and then open the Utility Properties as discussed in **Section 3.2**. The Utility Properties dialog box will open as shown below.



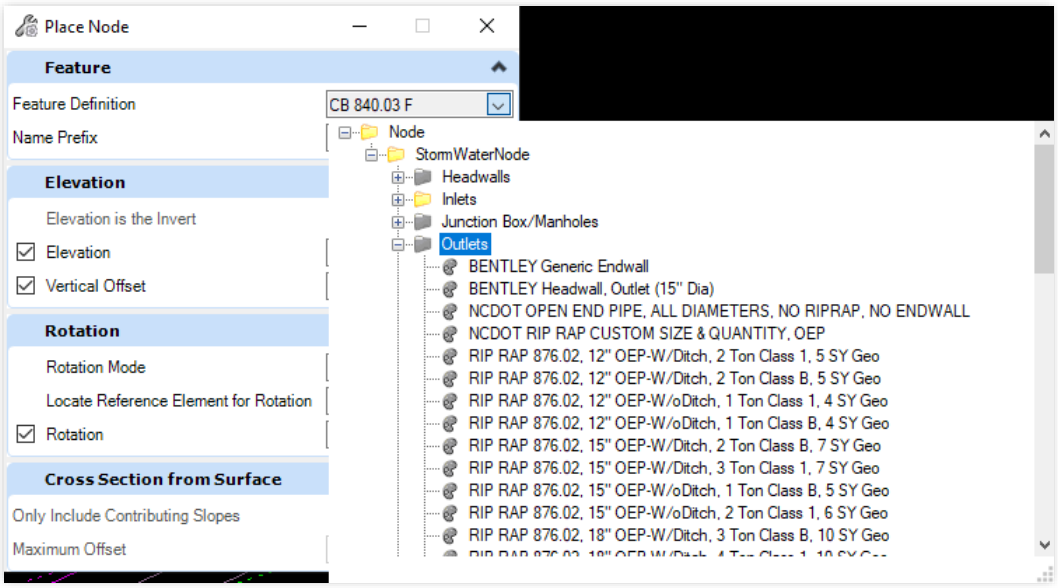
- Notice the "notes field" contains default instructions by NCDOT for the node.
- The default properties for the node are pre-populated from its catalog and prototype (explained in **Section 4.5** below).
- It is the responsibility of the user to change the applicable properties (catalog gutter, road cross slope, elevation, inlet location, etc.) depending on each specific node and circumstance.

4.3 Nodes: Outlets

Node Outlet Options and Guidance

Outlet placement is done with the same relative process as outlined in **Section 4.1**

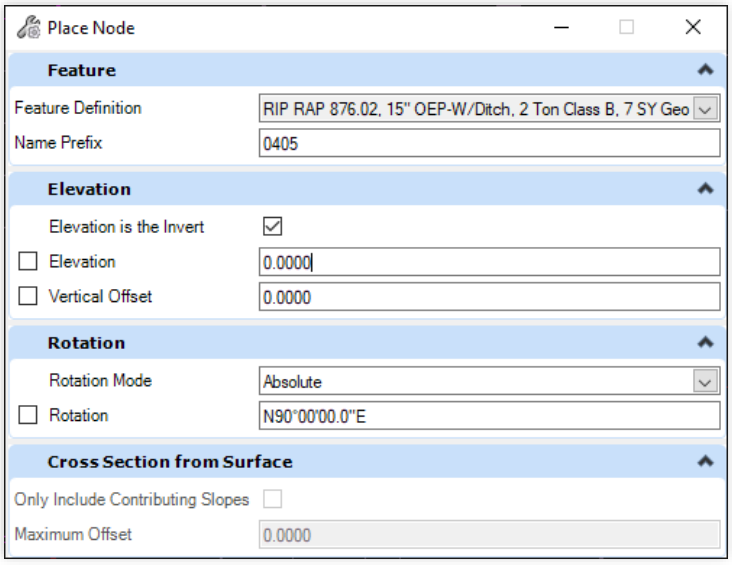
- The Place Node dialog box with the outlet node types is pictured below.



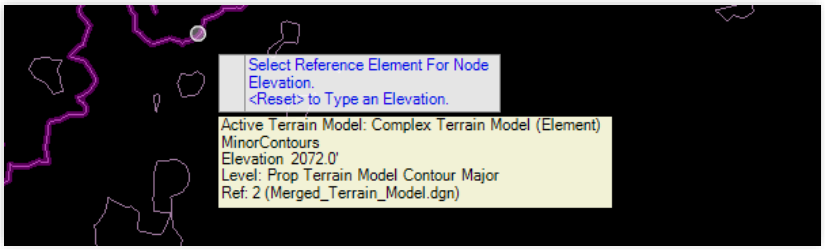
- Headwalls (in the headwalls folder shown in the screenshot above) can be placed as either an outfall node or an inlet node and the software will change them automatically depending on how pipe conduits connect them (see **Section 8.1** for conduit placement).

Helpful Hint: It is likely that the outlet pipe size is not known yet and will require unknown riprap size and/or endwall size. In this case, select the feature definition as *"NCDOT OPEN END PIPE ALL DIAMETERS, NO RIPRAP, NO ENDWALL"* then, once the system is completely designed with an appropriately sized outlet pipe, the user can change the feature definition using the properties tool.

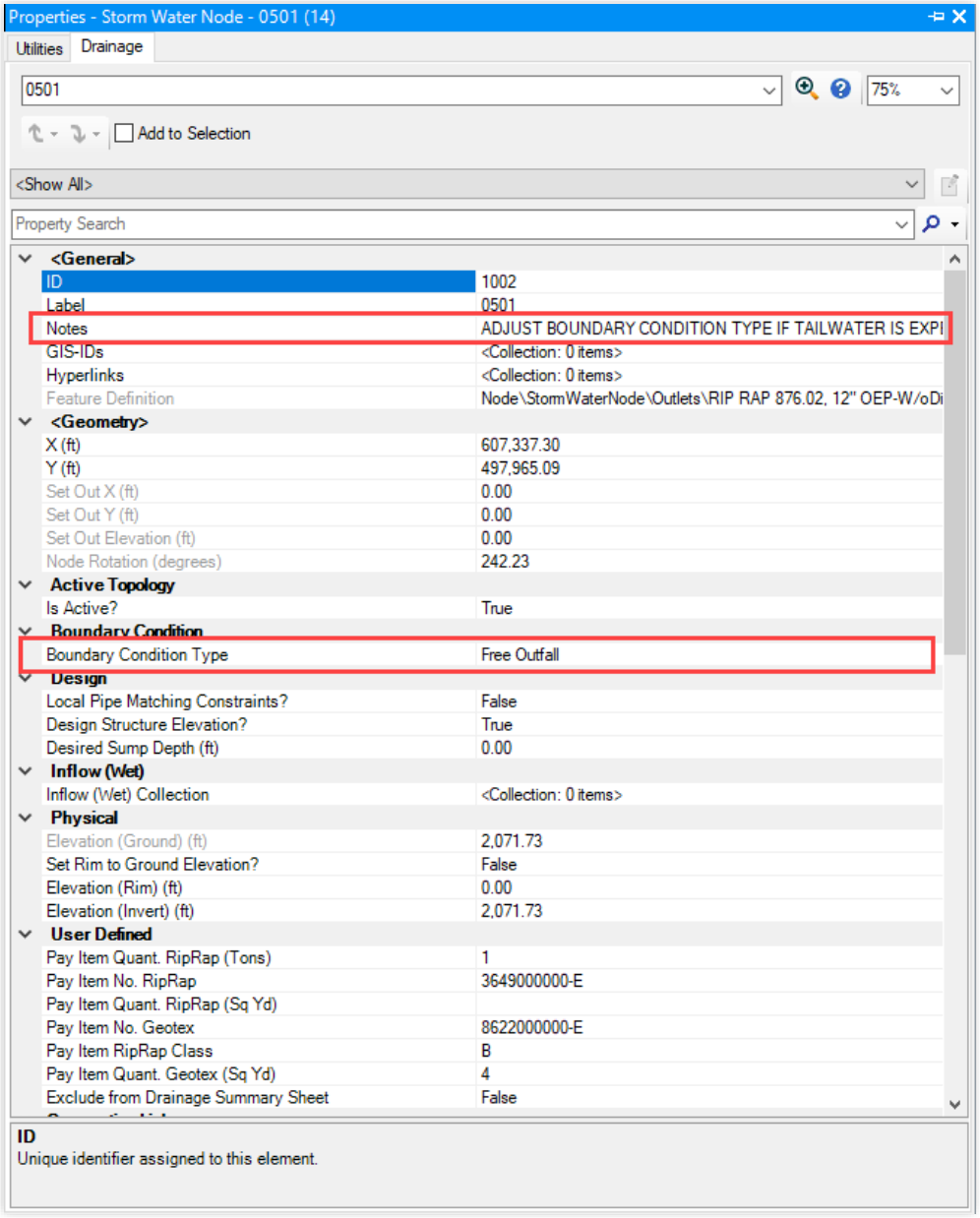
- Recommended settings for outlet nodes will vary from inlet nodes as shown below.



- The typical elevation reference for an outlet will be the existing terrain/merged terrain (or user input) as opposed to a roadway CMD element.

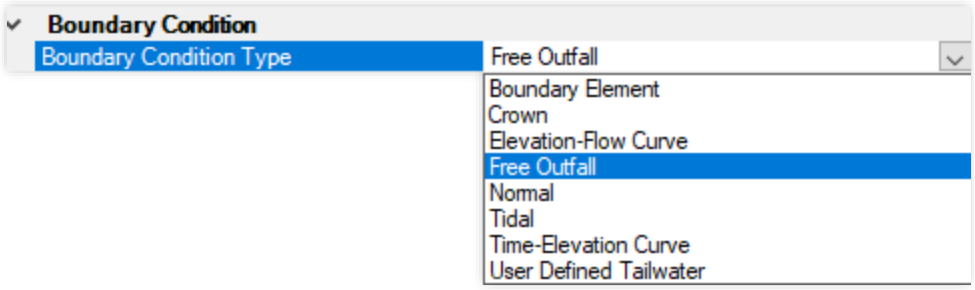


- After the outlet is placed and rotated properly, open the utility properties for the outlet node (see **Section 3.2**)



- Notice the notes field provided by NCDOT and change the boundary condition as necessary. Boundary condition selections are discussed more in depth on the next page.

- The screenshot below shows the boundary condition options.



Boundary Condition Option	Applicability / Description
Boundary Element	▪ *This will rarely be used. It allows the user to select a Drainage and Utilities element that will receive the outfall discharge.*
Crown	▪ *This will rarely be used. This sets the tailwater elevation at the crown of the outfall conduit.*
Elevation-Flow Curve	▪ *This will rarely be used. It allows the user to define tailwater elevations based on the flow being discharged.*
Free Outfall	▪ ***This should be the default**. It allows the program to select the appropriate depth depending on the flow regime – critical depth for subcritical and normal depth in the pipe for super critical.*
Normal	▪ *This will rarely be used. Normal in this case means the normal depth of the upstream conduit before the outlet.*
Tidal	▪ *This will rarely be used. It allows the description of elevation changes over time. For coastal projects, users should use the most conservative tide elevation and the "User Defined Tailwater" option.*
Time-Elevation Curve	▪ *This will rarely be used. It allows the user to define tailwater elevations based on time.*
User Defined Tailwater	▪ *This will be used regularly when boundary conditions exist with a known elevation that will not vary. It allows the user to enter an elevation used for the tailwater.*

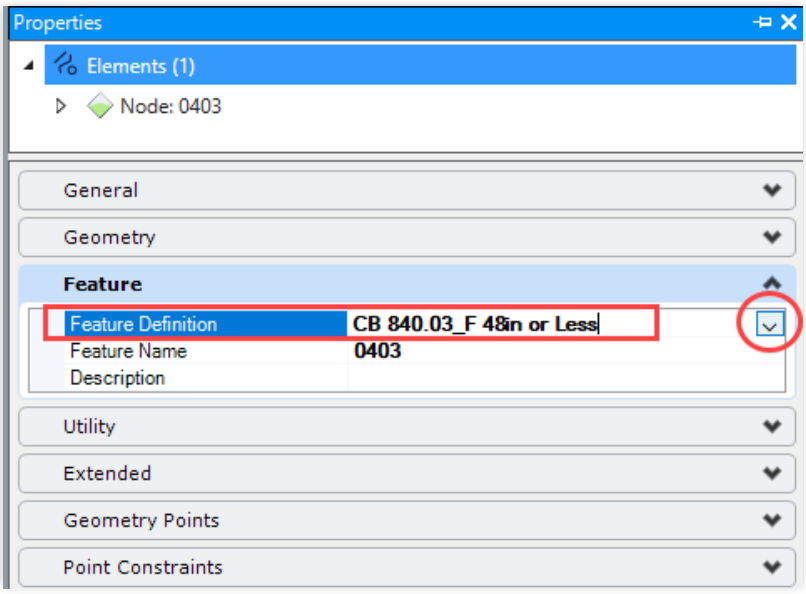
4.4 Nodes: Adjustments

Changing Structure Type or Adjusting Elevation After Placement

Situations may arise where a node type must be changed from one to another or rim/top elevations edited. The steps below outline the basic procedures to edit these properties.

To change a node's type after placement, follow the steps below.

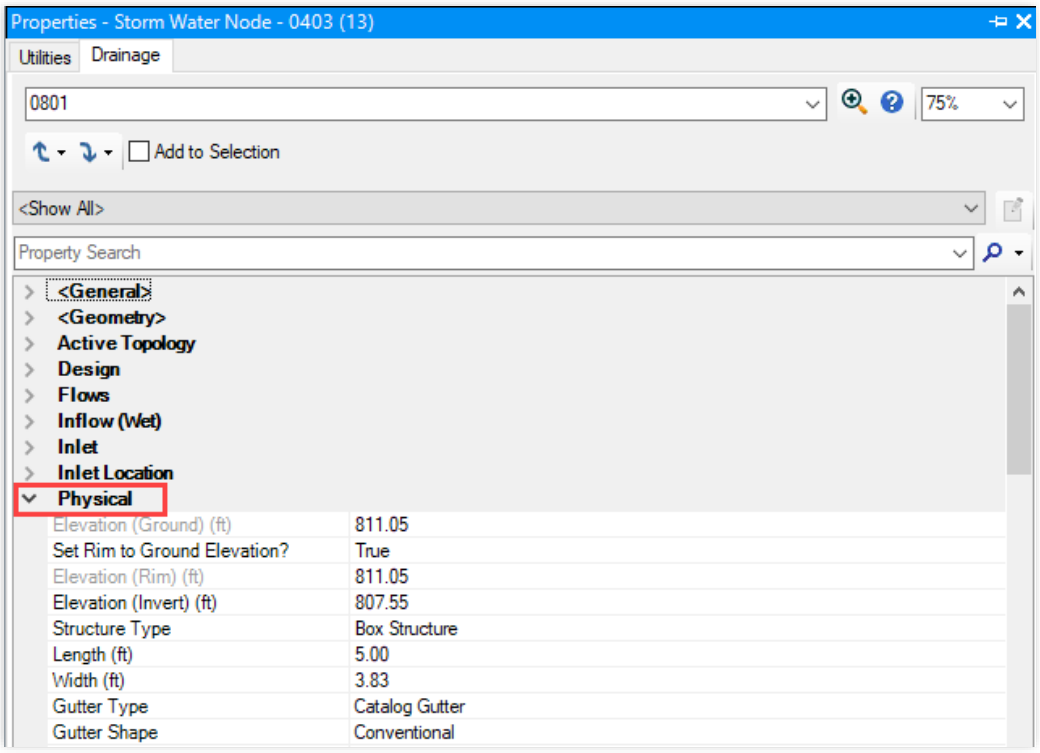
- Select the node and open its properties. Locate the feature definition drop down in the properties as shown below.



- Use the drop down (circled in red above) to select the new node type.
- The node type has now been changed. Update elevations and other characteristics accordingly depending on the type of node.

To edit a node's elevation after it has been placed follow the steps below.

- Select a node, open its utility properties, and navigate to the "physical" properties tab (see screenshot next page.)



- Rim elevations can be changed under the Physical properties as shown in the screenshot above

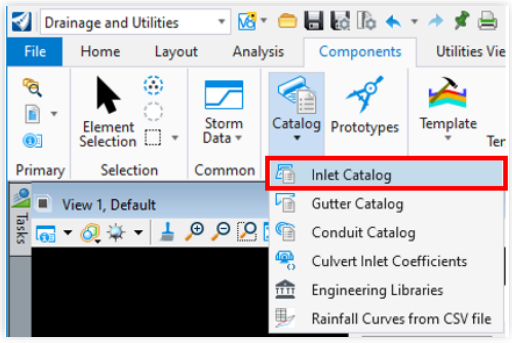
- Notice that in the screenshot above *Set Rim to Ground Elevation?* is set to true. When this is set to true **and** the node's elevation is referenced to a terrain or roadway CMD element, the rim elevation of the node cannot be changed. In this scenario, the elevation of the node will be automatically updated when the CMD element or terrain is updated. (note: the proposed terrain will automatically update with the CMD it is referenced to - see note at end of **Section 2.5**)

- If the user still wishes to change the elevation, *Set Rim to Ground Elevation?* should be changed to false and the *Elevation (Rim) (ft)* field can be now be edited manually. **Note:** This unlinks the rim elevation from the associated terrain model or CMD element. When the terrain or CMD element's elevation is updated, the *Elevation (Ground) (ft)* will change automatically, but the *Elevation (Rim) (ft)* will remain as what the user has input.

4.5 Nodes: Background Data

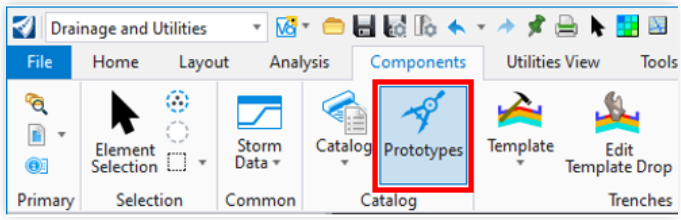
Miscellaneous Inlet Components (for Information Purposes Only)

- The Inlet Catalog is where default grate type, grate size, and structure size are housed. It is located at Components Ribbon Tab > Catalog Drop Down > Inlet Catalog



  - **The catalog should be used for information purposes only. Editing it should be done at the users own risk. It is not recommended to edit the default NCDOT inlets. In rare situations, inlets can be duplicated, and new ones created.**
  - **If any commonly used inlet is missing or errors are found in catalog items please refer to Section 16.1 to contact NCDOT**

- Additional Node/Inlet properties and types are also stored in the Prototypes library located at the Components Ribbon Tab > Prototypes



  - **The Prototypes library should be used for information purposes only. Editing it should be done at the users own risk.**

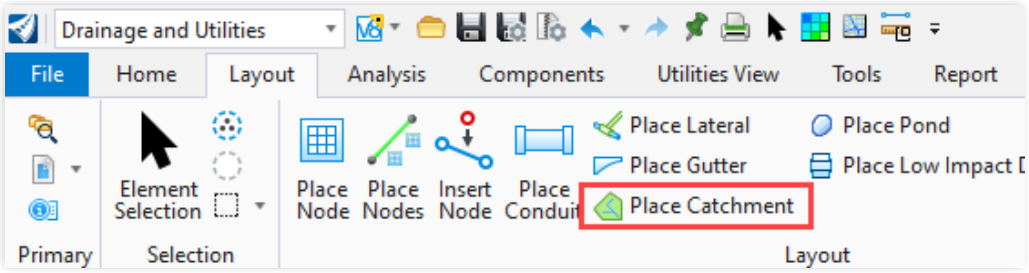
Helpful Hint: A drainage **Feature Definition** (such as a CB 840.03 G) is typically a combination of a prototype, catalog (if applicable) and a graphic cell.

5.1 Catchments: Placement

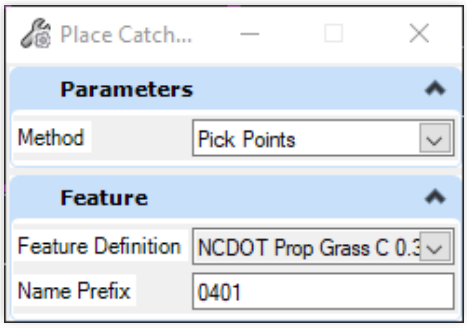
Placing Drainage Areas and Assigning Them to Nodes

ORD refers to drainage areas as "catchments". Catchments are placed by drawing shapes. They must be placed using the workflow described below.

- To place catchments, go to the Layout Ribbon Tab > Place Catchment



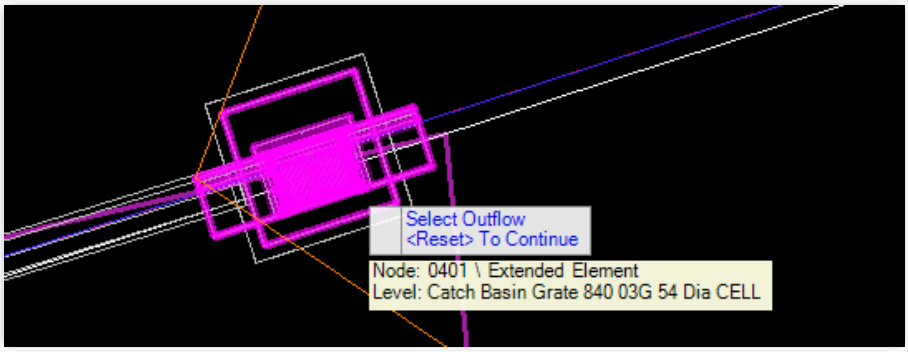
- The Place Catchments dialog box will open



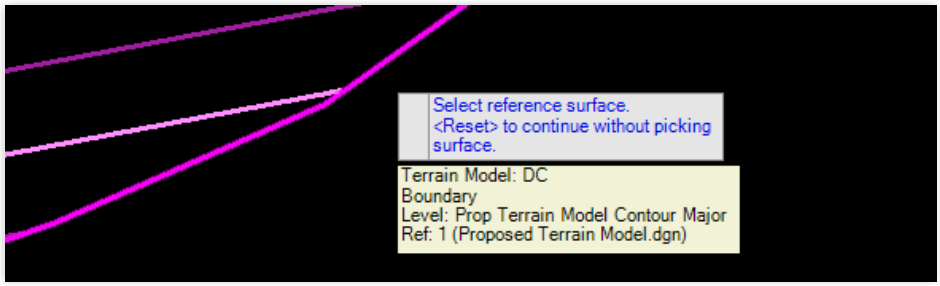
Place Catchment Option	Description
Method	▪ *Three options: Pick Points, Pick Shape, Flood Fill*
Feature Definition	▪ *This is where the land use will be selected. At this time only one C-Value can be selected however, the C-Values can be edited individually for each catchment after placement (see Section 5.2 below). If the C-Value will be edited individually it does not matter which feature definition is chosen, but it's recommended a logical one be chosen.*
Name Prefix	▪ *This is where the user will name the catchment (name should correspond to node associated with the catchment. NCDOT typical structure and area numbering should be followed (for example this node and catchment are named 0401)*

- After the user selects points or a pre-drawn shape as the catchment, the option to select outflow will appear. This is where the node associated with the catchment will be assigned. Hover over the node to assign till it is highlighted as shown below.

Helpful Hint: Right click is the same as <reset> to continue without selection



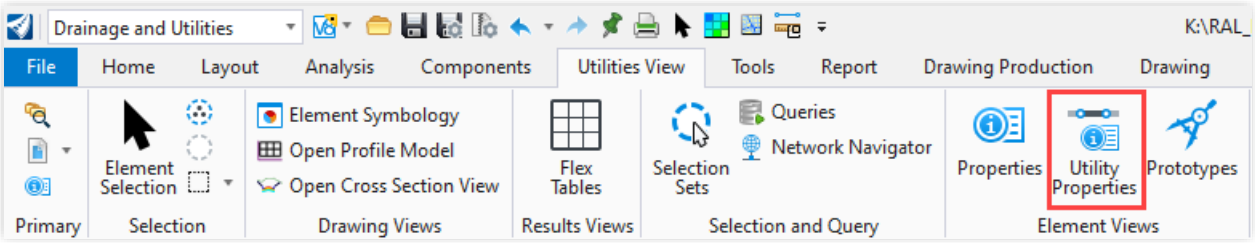
- The next step is to select the terrain associated with the catchment. Hover over any part of the terrain and select it as shown below. If the user prefers to not assign a terrain, they can simply right click. A terrain can be assigned later or changed if needed. Assigning a terrain will not affect the design but will allow the drainage area to be viewed in 3D (**Section 13.1**).



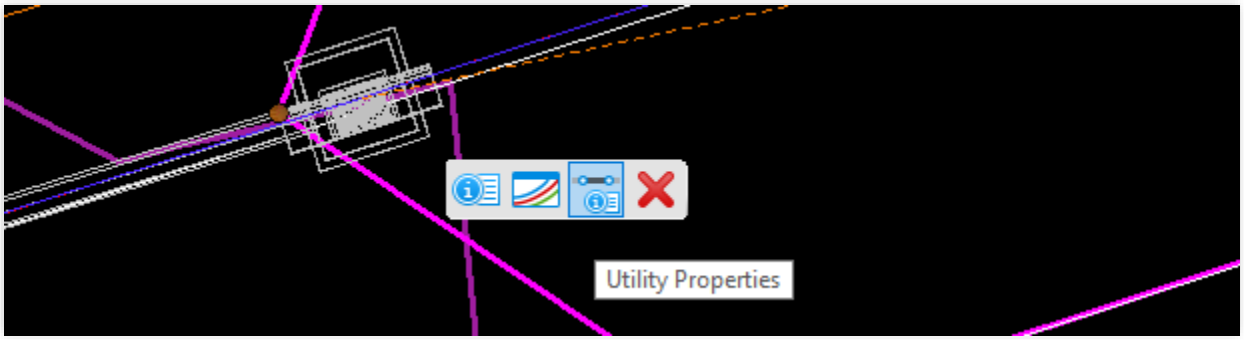
5.2 Catchments: Time of Concentration and C-Values

Using the Utility Properties to Set the Time of Concentration and C-Values

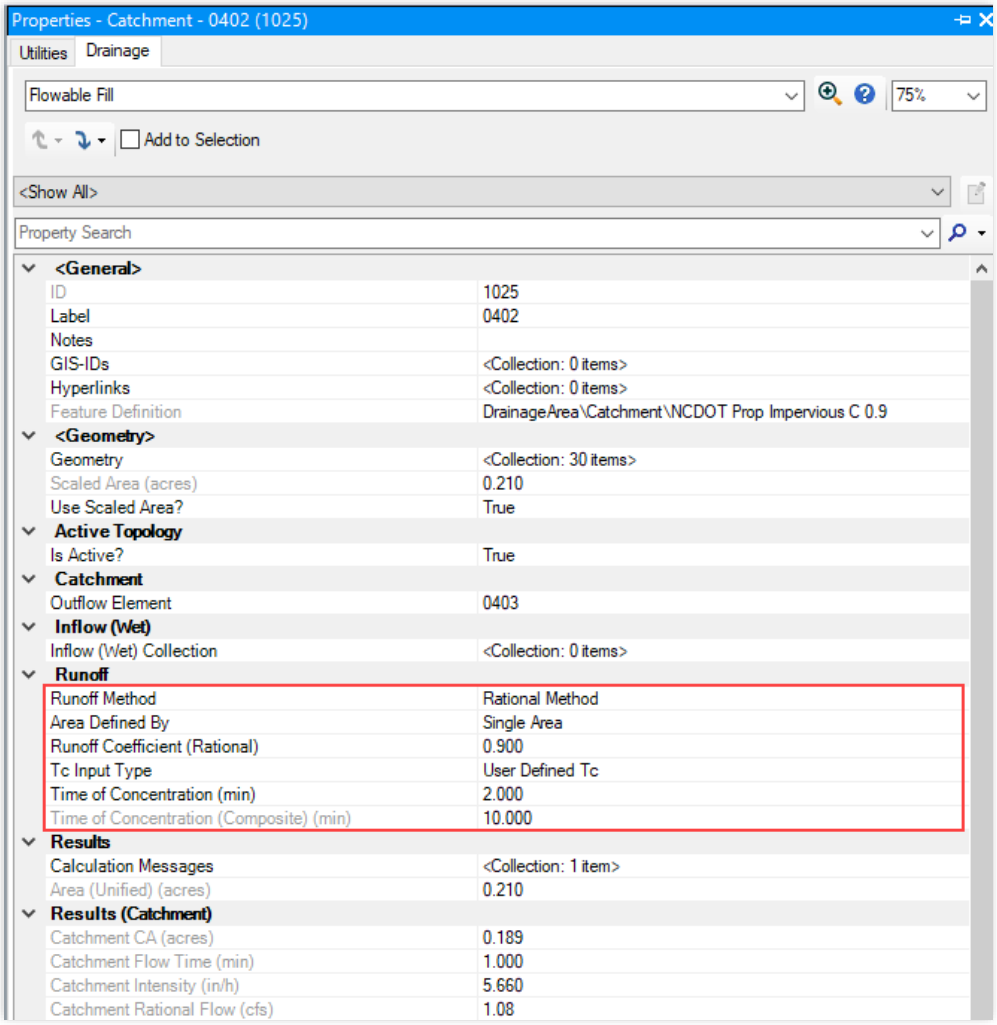
- To set the time of concentration and C-values for a catchment, select the area and open the Utility Properties under the Utilities View Ribbon Tab (**Section 3.2**)



- The catchment's Utility Properties can also be accessed by selecting the catchment and hovering over the catchment until the quick toolbar button appears as shown below.



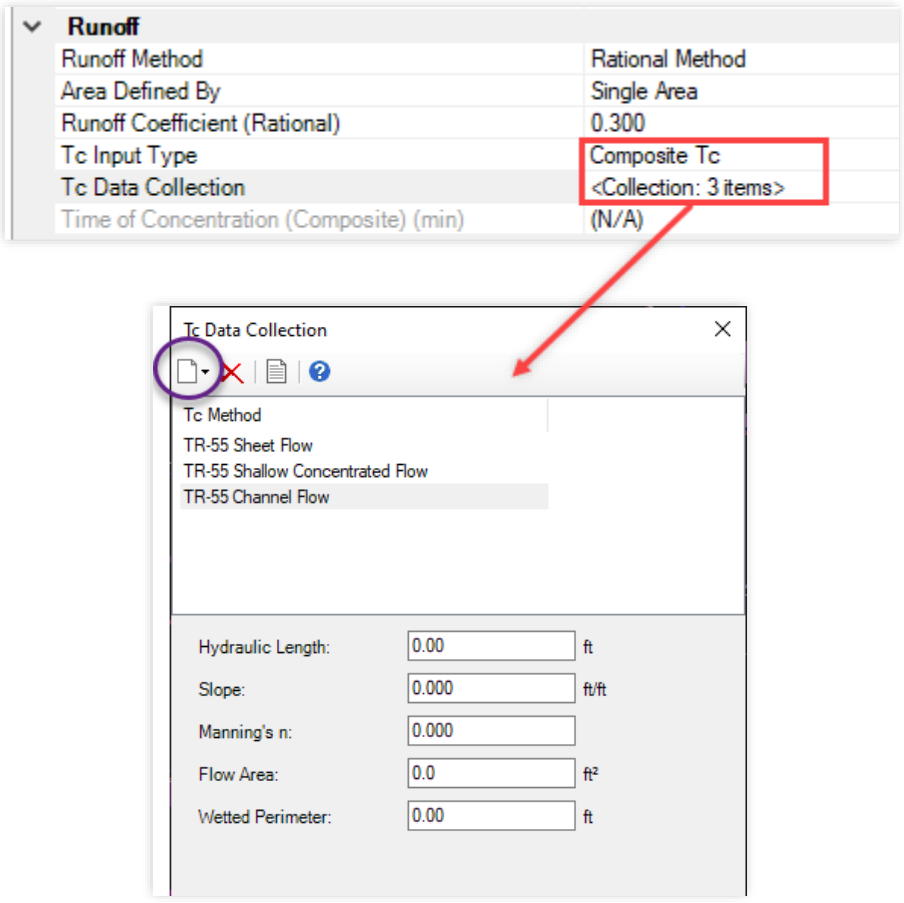
- Click the utility properties button and the catchment's utility property's dialog box opens.



- Time of Concentration and C-Values are edited in the runoff category outlined in red above. **NOTE: The user should not change the runoff method from the default rational method for standard NCDOT storm drainage systems**. More complex methods are not outlined in this document – user should only use other methods after consultation with NCDOT reviewer.
- Even though the minimum time of concentration is 10 minutes, the time of concentration for catch basins with small, impervious drainage areas should be set to a more realistic number (typically a very small number such as 2 minutes). The more realistic (smaller) time of concentration will only be used for calculating the accumulating system time and using a small number will prevent the accumulating system time from going above the 10 minute minimum before it should. The flows

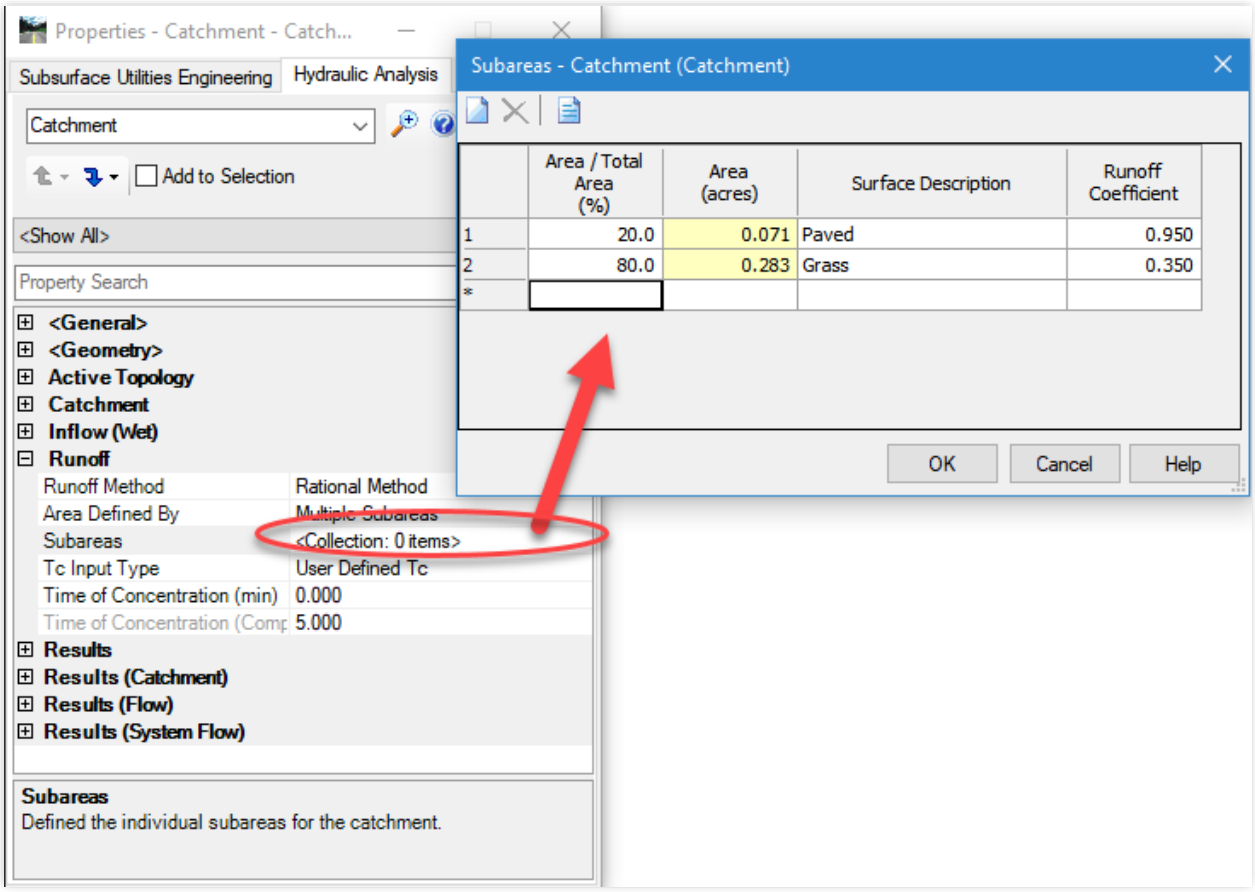
that are calculated and used for pipe sizing will be based off of the minimum 10-minute rainfall intensity values even though the more realistic,entered, time of concentration is less than 10 minutes.

- Time of Concentration can also be broken up into multiple calculations instead of being user entered as shown below. To add the different collection items as shown below, use the "new" button circled in purple.



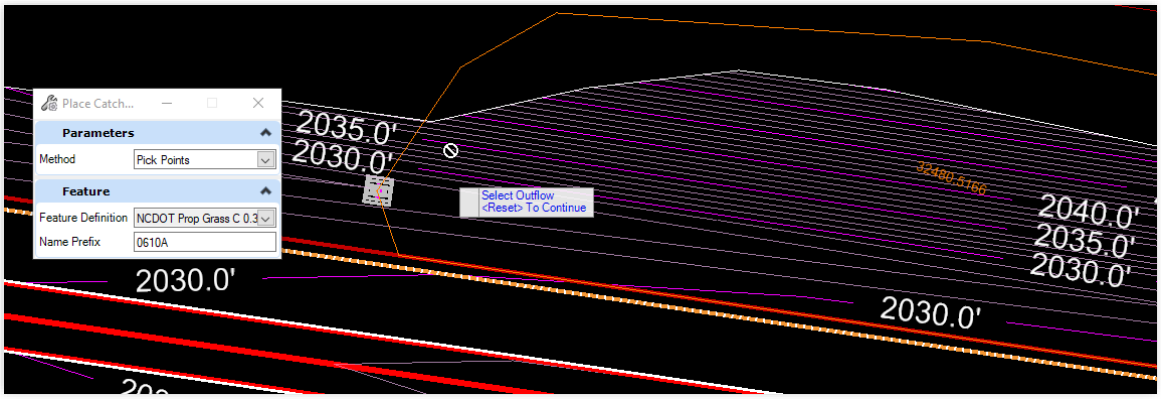
- Many different time of concentration methods are available to use. NCDOT recommends the TR-55 method as shown above.
- In most cases the runoff coefficient will need to be weighted based on the relative percentages of different land uses within it.
- There are currently no tools for automatically determining the percentages of land use based on global land use shapes. This feature may be available in late 2021 and would be covered in an updated release of this guide at that time.

- The simplest method for a weighted runoff coefficient is to use the "*Area Defined By*" Input field (shown in the screenshot below). It can be changed to multiple areas as shown below to produce a weighted C-value.

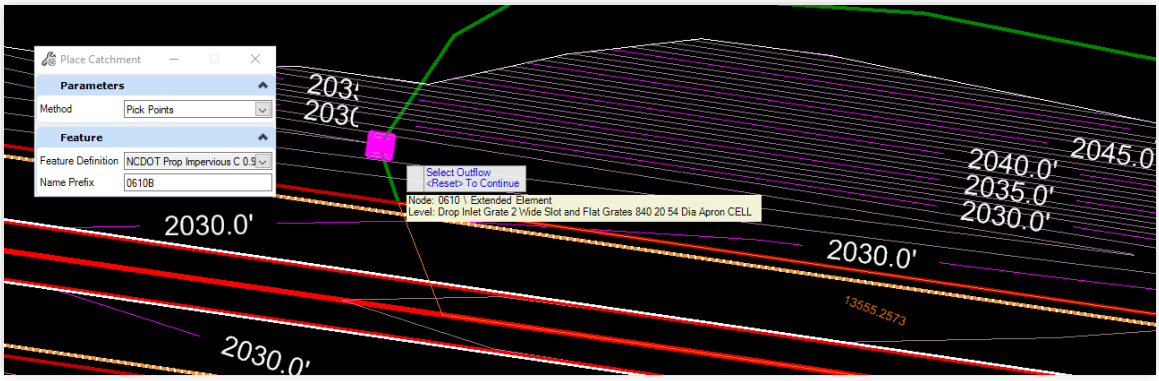


- If the user does not prefer the option above, another option to calculate a composite C-Value is to assign multiple catchments to one node. ORD does not have a limit on how many drainage areas/catchments can be sent to a single node. See below for the standard practice to assign multiple drainage areas with different land uses to a single node.

- Place the first catchment (**Section 5.1**) in the first land use sub-area. In this example, the first area will be the grassed area draining to the 2GI in the ditch and will use the "NCDOT Prop Grass C 0.3" feature definition. The name of the catchment should be the name of the node followed with A, B, C, etc. depending on how many catchments the user is assigning.



- After the first catchment is placed, create another catchment for the next land use. In this example, the proposed roadway draining off the paved shoulder and to the 2GI in the ditch will use the "NCDOT Prop Impervious C 0.9" feature definition. Note the name of the second catchment, "0610B"



- Once all catchments have been placed and the system/inlet analysis is run in **Section 7.1**, the software will automatically calculate the composite C-value and use it for the inlet and system calculations. However, each catchment will still retain its original C-value and peak flow when its utility properties are viewed separately.

Helpful Hint: Multiple catchments can be useful in rare situations where flow from an impervious sub-area calculated by itself is higher than the flow from the entire, larger area with a weighted C-value. This occurs when a smaller wooded/grass sub-area with a high time of concentration is weighted with a large impervious sub-area with a low time of concentration. Drainage and Utilities does not account for using the higher flow however, when multiple catchments are used as outlined above, each catchment's flows can be checked to verify if this situation is occurring. If it is, then the catchments can be manipulated accordingly to represent only the highest flow from the impervious sub-area.

6.1 Gutters: Overview

Explanation of Gutter Capabilities and Purpose

Gutters in the Drainage and Utilities model workspace are slightly confusing because they are separate from the roadway gutter linework/elements. For NCDOT projects, the main (and required) purpose of these gutters is **to assign bypass flow** from an upstream inlet to a downstream inlet. Gutters placed in the Drainage and Utilities model represent **only a hydraulic connection between nodes**. They are not to be confused with and are not part of roadway's physical gutter design in the 2D/3D model.

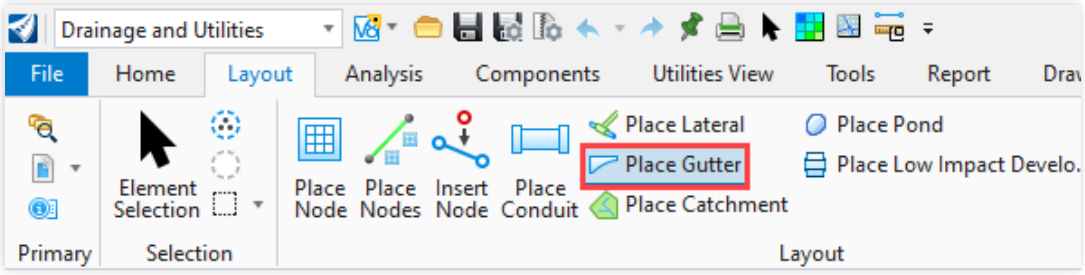
Other than using gutters to assign bypass, they also can be used to analyze spread. This feature is mostly redundant because the spread of interest is typically taken at the inlet and not at random places along the gutter. **At this time, it is recommended that gutters be used mainly for purposes of assigning bypass and the inlet computations used to check spread.**

6.2 Gutters: Placement

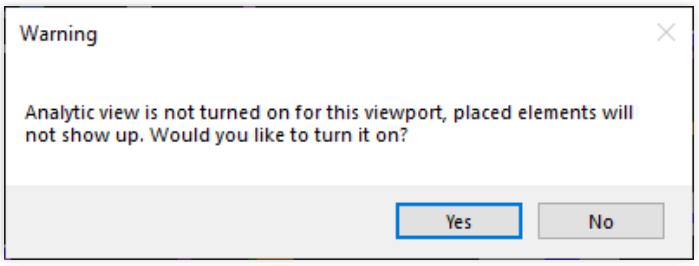
Using the Gutter Placement Tool to Connect Nodes

As mentioned in the section above, bypass is assigned in ORD by placing a gutter connection between two inlets.

- To place a gutter connection and assign bypass, go to the Layout Ribbon Tab > Place Gutter

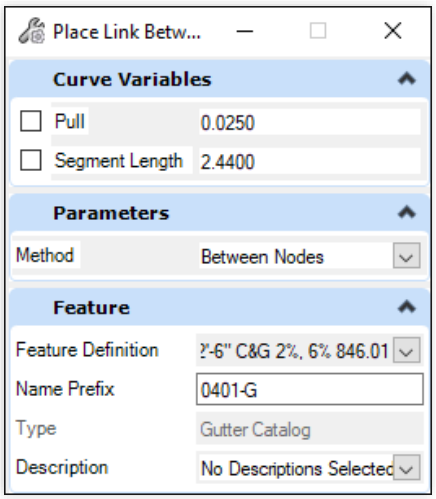


- A warning message may pop up below to turn on Analytic view. Click yes to turn on Analytic view. Gutters can only be viewed in analytic view.



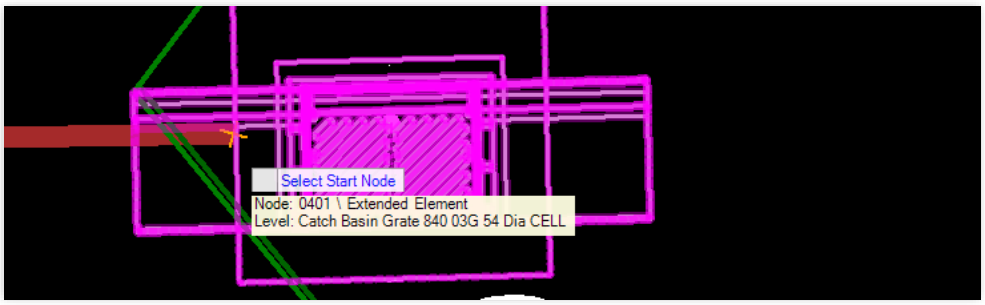
Helpful Hint: Analytic View is also used to turn on and off labeling/ sets of labels for drainage elements. This labeling is only a visualization and not a permanent / moveable text element. For more clarification on this process, **Section 12.2** covers this labeling and sets of labels in more depth.

- The Place Gutter dialog box will open (see next page)



Place Gutter Option	Typical User Selection
Curve Variables	▪ *Leave unchecked*
Method	▪ *NCDOT users should select between nodes*
Feature Definition	▪ *Use the drop down to select the C&G type that connects the elements.* ▪ *If there is a super elevation transition where flow in the gutter would cross the road to the next inlet use the feature definition "NCDOT Super Transition." This is a generic gutter section that should only be used for bypass assignment purposes.* ▪ *A gutter with a negative road cross slope or gutter cross slope cannot be used (system will not run).*
Name Prefix	▪ *Name the gutter connection after the upstream node. Since gutters are a type of conduit you will have to add a "-G" after the name so that it does not conflict with the pipe conduit names.*
Description	▪ *No descriptions are available at this time, leave as default*

- After the selections are made above, select the start node for the gutter placement. Note that there are specific snap points on each side of the node which will show up as orange crosshairs.



- The user can either snap directly to the downstream node to assign the bypass or hit (not hold) CTRL to place bends in the gutter connection (not recommended unless gutter is curved).
- Select the downstream node for the gutter connection and the gutter linework will appear. Note: It may be difficult to see the gutter linework as it is usually plotted directly on top of pipe connections. Turning on and off appropriate levels when necessary can fix this problem so that the gutter can be seen/selected.
- Gutter properties are viewed the same way that node properties are viewed as outlined in **Section 3.2** (properties and utility properties).

6.3 Gutters: Advanced

Complex Gutter Sections and Spread at Intervals

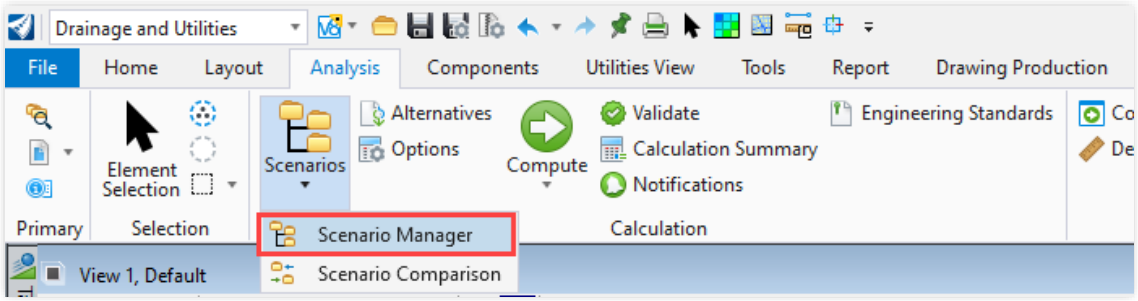
More advanced features of gutters include using "Gutter Sections" for varying geometry gutter sections (i.e. super elevation transitions) or for viewing spread incrementally along a gutter rather than just the start and end. This requires a proposed terrain and the software will automatically cut gutter sections (cannot be user defined). At this time, due to the lack of flexibility and some bugs encountered, the "Gutter Sections" feature will not be outlined in this document. Future versions of this manual will incorporate these advanced tools as they become more reliable and useful on NCDOT projects.

7.1 Spread & Inlet Computations: Running Scenarios

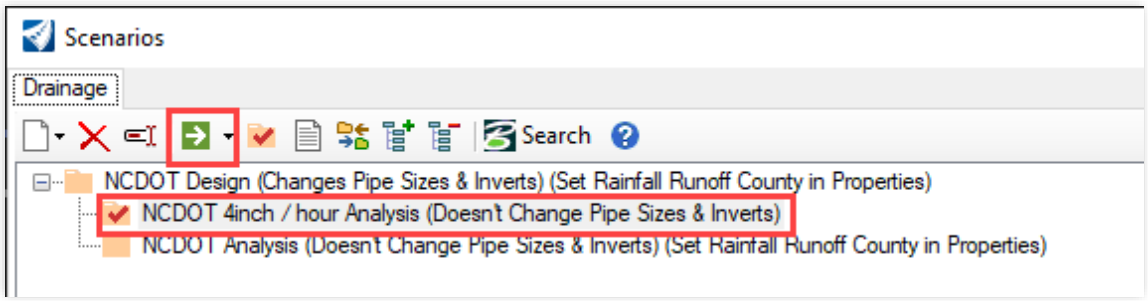
Using the NCDOT 4.0 Inch/Hour Scenario to Calculate and Analyze Spread

Before adding pipe conduits and running an entire system, it is recommended to run the spread analysis and make any necessary changes to inlet locations. To run the 4.0 inch/hour spread analysis, follow the steps below.

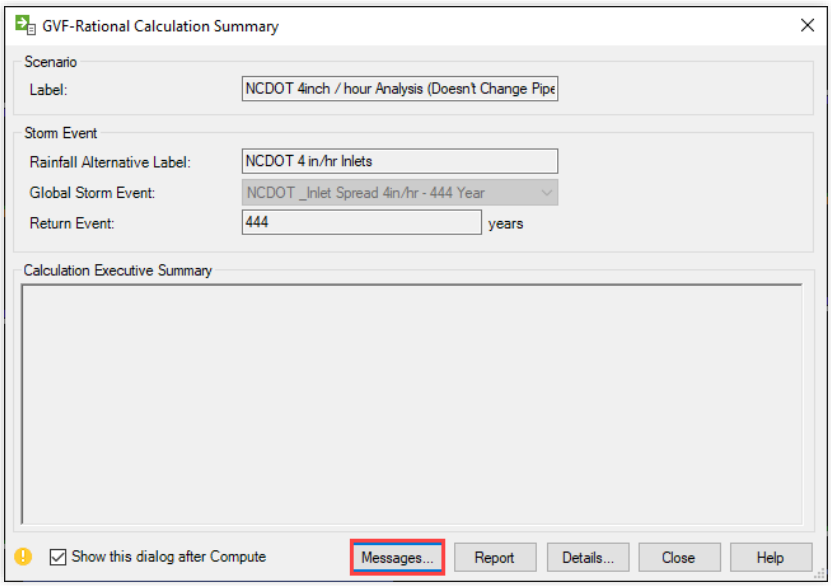
- Under the Drainage and Utilities workflow, navigate to the Analysis Ribbon Tab > Scenarios > Scenario Manager



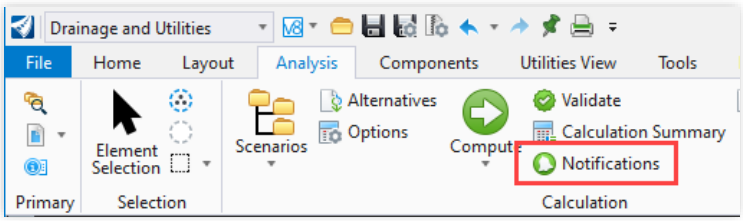
- The scenarios dialog box will open. Highlight the **NCDOT 4 inch / hour Analysis** and click the green arrow to compute



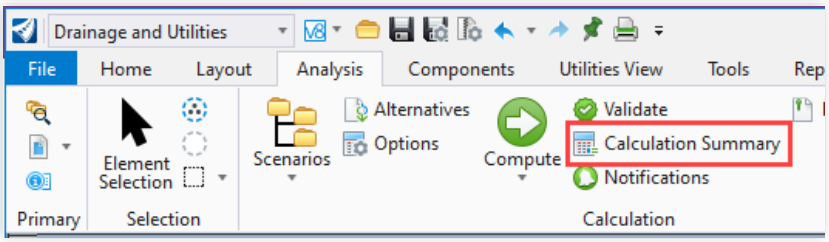
- The calculation summary dialog box will open after the calculations are complete. Warnings and errors can be viewed by clicking the messages button outlined in red below



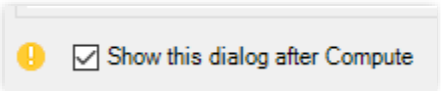
or by selecting the notifications option as shown in red below



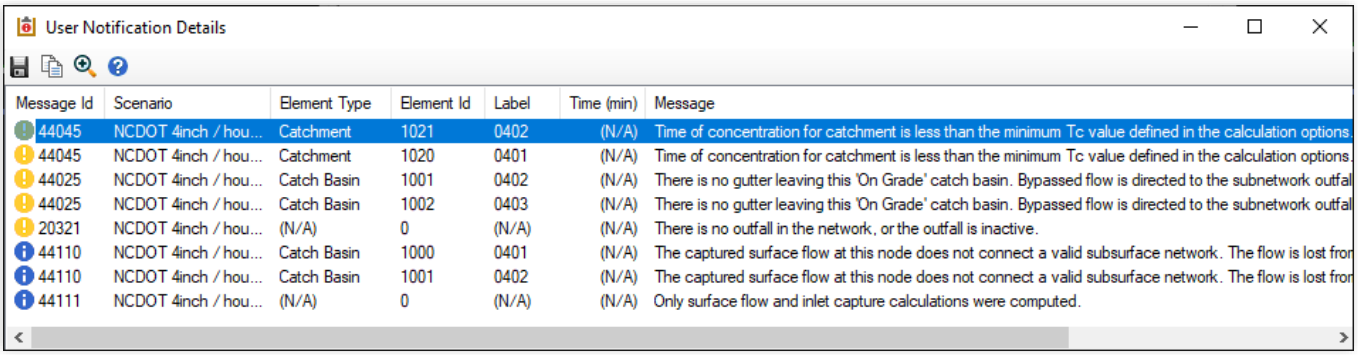
- Note: If the "calculation summary dialog box" does not open automatically after the "compute" is run, open it by clicking the option in the screenshot below



- The box that states "Show this dialog after Compute" can be checked so that the dialog box appears after every compute.



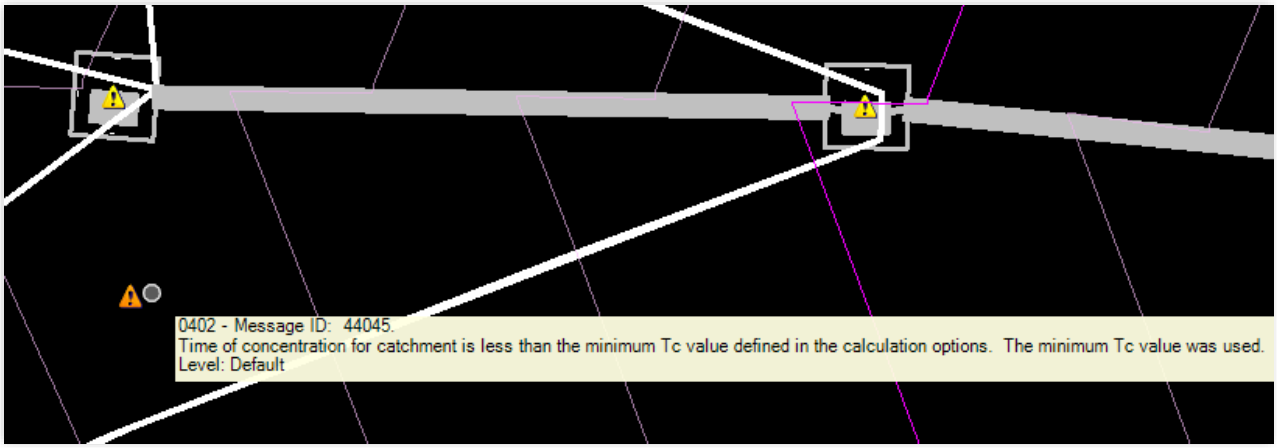
- There will be warnings and errors since the system is not yet complete. Typical warnings and errors at this step in design are shown below.



Helpful Hint: If an error or warning message is double clicked it can automatically take the user to where the error/warning is occurring

In addition to the warnings and errors shown in the "User Notification Details" dialog box above, there will also be warning notifications that appear in plan view that resemble yellow yield signs (see screenshot below next bullet).

- The user can hover review the design in plan view and hover over one of these "yield" symbols until a description of the warning/error is displayed next to the cursor. This can be helpful when a user has many systems/nodes and needs to review errors/warninga in one specific area of the design in plan view.

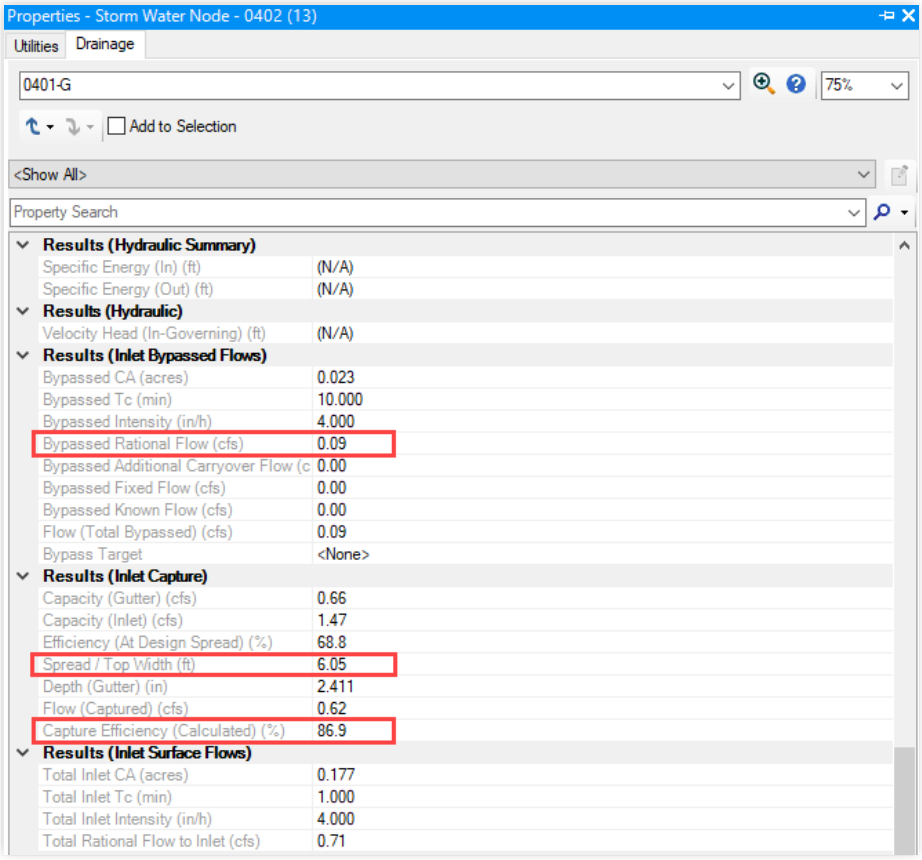


7.2 Spread & Inlet Computations: Viewing Results

Using the Utility Properties to View Inlet Computations

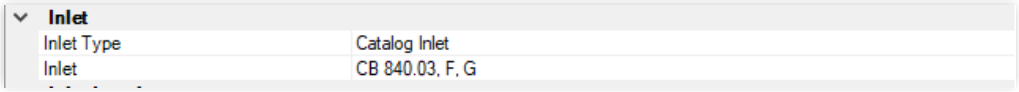
After the NCDOT 4 inch / hour scenario has been run, typical inlet computations such as spread, bypass and inlet efficiency can be checked by viewing the Utility Properties.

- Select a node and open the utility properties (**Section 3.2**) Scroll down to the results section in the utility properties window to view calculation results.



- Spread, bypass, inlet efficiency and many other calculations are shown.
- Users may also prefer to check spread calculations by using summary tables called Flex Tables (guidance on Flex Tables is outlined in **Section 10.1**)
- For sag inlets, spread left and right will need to be verified with another software and reported in the final computations package.

Helpful Hint: If bypass flow and inlet efficiency are not being computed, the user may need to scroll up and change Inlet type from 100% efficiency to Catalog inlet (see below)

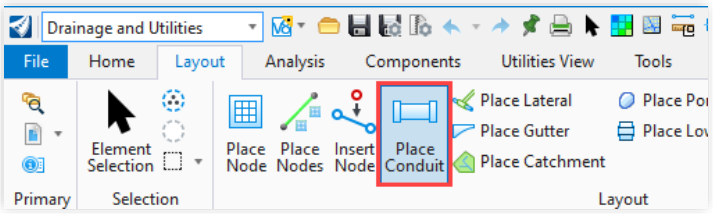


8.1 Conduits: Placement

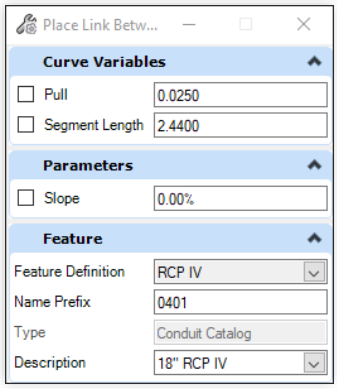
Using Conduits to Connect a System

After the inlets are appropriately spaced and the designer is confident in the layout, the nodes will be connected using conduits. In ORD, conduits will mainly be used for pipe connections. Conduits may also be used as channel connections; however, this section will focus solely on pipes.

- Under the Drainage and Utilities workflow, navigate to the Layout Ribbon Tab > Place Conduit



- The place conduit dialog box will open

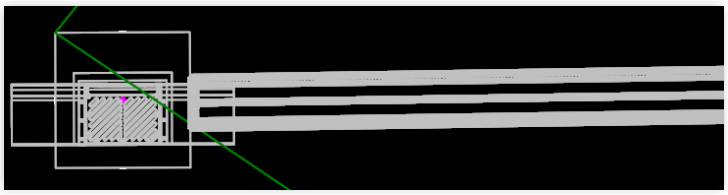


Place Conduit Option	Typical User Selection
Curve Variables	▪ *Leave unchecked*
Slope	▪ *If a design slope is desired, this box can be checked. When you are placing the conduit, the slope in this field will automatically calculate based on the node bottom elevations.*
Feature Definition	▪ *Use the drop down to select the type of pipe (channels are also an option)*
Name Prefix	▪ *Name the conduit connection after the upstream node (ex. 0401)*

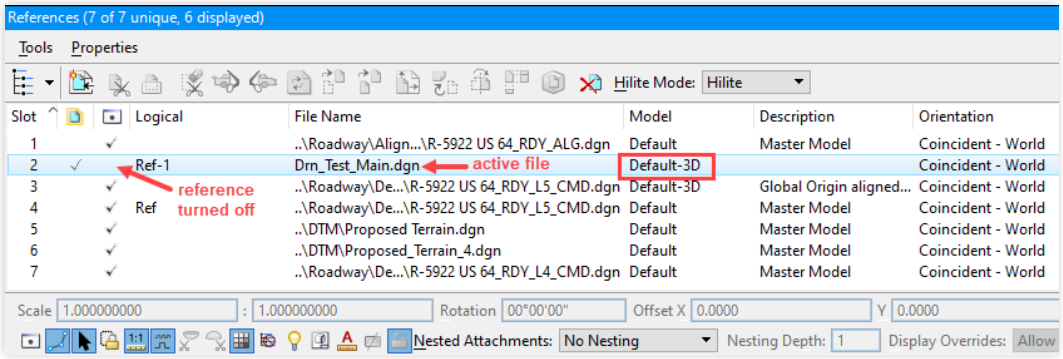
Description	▪ *This is where pipe size will be selected* ▪ *Helpful Hint: The software will design pipe sizes for the user at a later step so a generic size (i.e. 15-18") can be chosen initially for all conduits.*

- After the selections are made above, select the start node for the conduit placement. Note that there are specific snap points on each side of the node which will show up as orange crosshairs.

- Select the downstream node and the conduit link will appear.



- Both a 3D and 2D element will be drawn. The 3D elements are stored in a separate reference of the active .dgn as shown below. The 3D reference should be turned off for this view. A second view can be opened for the 3D model space (see **Section 13.1**)

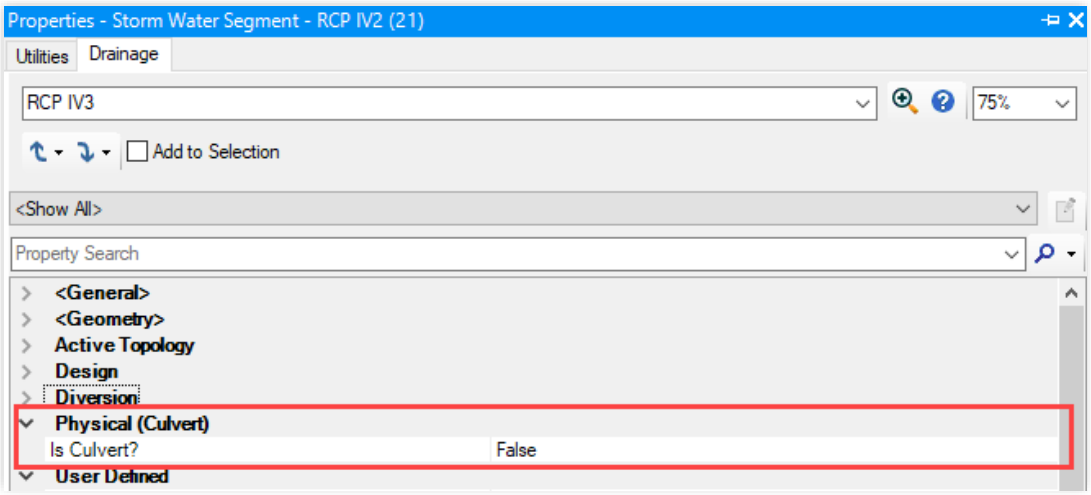


Helpful Hint: If a conduit is placed with one material and later needs to be changed to another (example: RCP to CMP), the n-value may not update and should be double checked and changed by hand if needed.

8.2 Conduits: Cross Pipes

The "Is Culvert" Property and Procedure for Cross Pipes

Users may notice that conduits have the option to be modeled as a "culvert" (see screenshot of utility properties for a conduit below). While this option may seem applicable to cross pipes (open end to open end), open end inlets and pipes with a headwall inlet, **current NCDOT guidance is for this to be set to false for all conduits**.



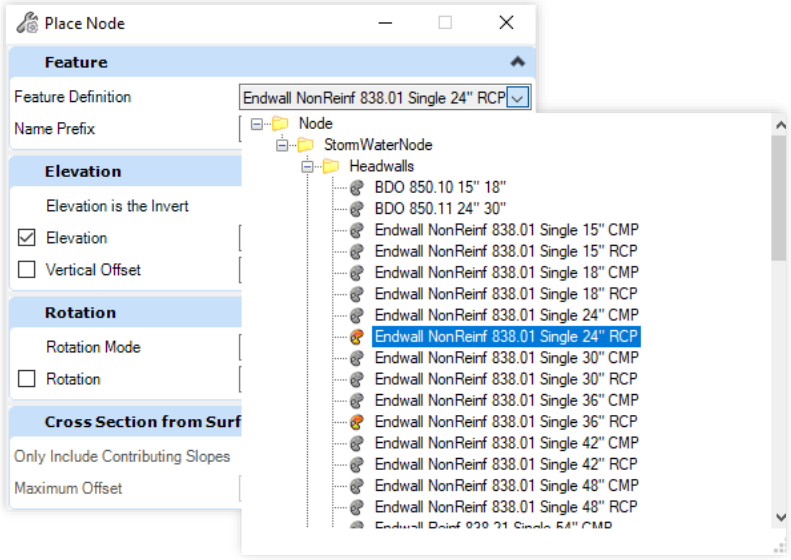
- Although this option allows a conduit that is a cross pipe to be modeled with inlet coefficient losses and also checks for inlet/outlet control, users should model cross pipes according the latest version of the NCDOT Guidelines for Drainage Studies and Hydraulic Design. A cross pipe can still be input in the drainage and utilities model (for quantities or to connect to a median inlet), however, separate calculations shall be provided as noted in the guidelines.

8.3 Conduits: Open End Pipes / Headwalls

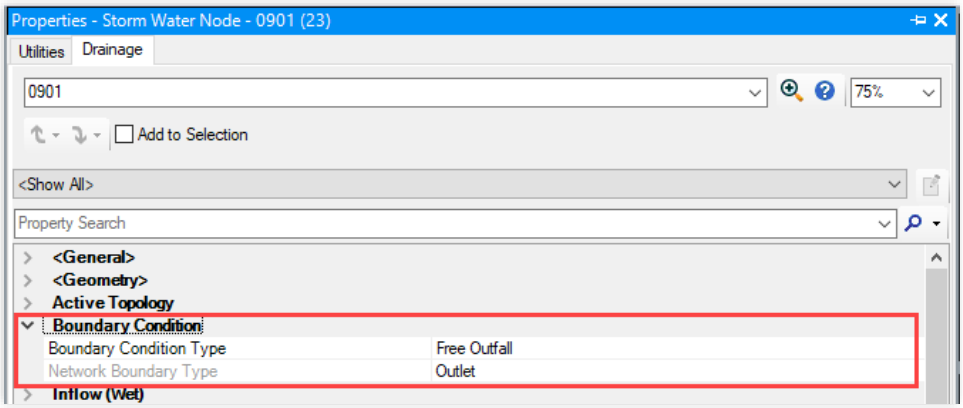
Procedure for Placing Headwalls as an Inlet Opening

As mentioned in **Section 4.3**, the headwall feature definitions housed within the node folder can be used as either inlet nodes or outlet nodes. The steps below cover special steps needed for open end pipes and pipes with a headwall.

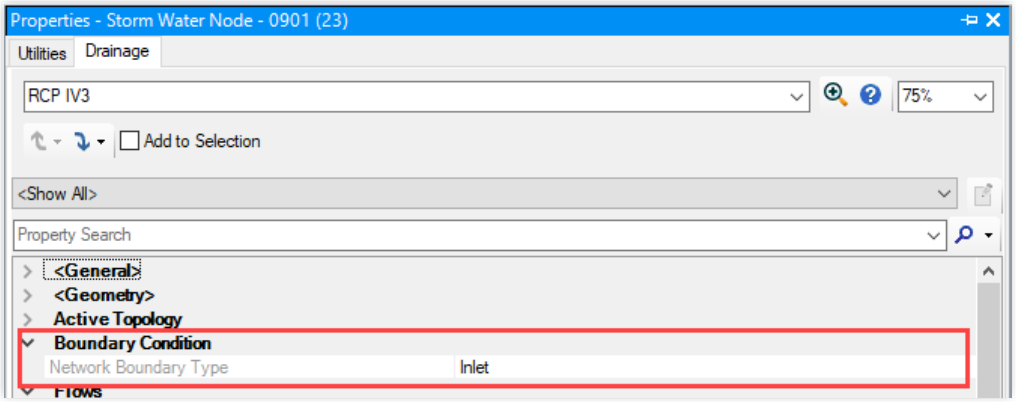
- Follow the steps in **Section 4.1** (Node Placement), but choose a headwall feature definition or open end (OEP) definition (headwall selections shown below)



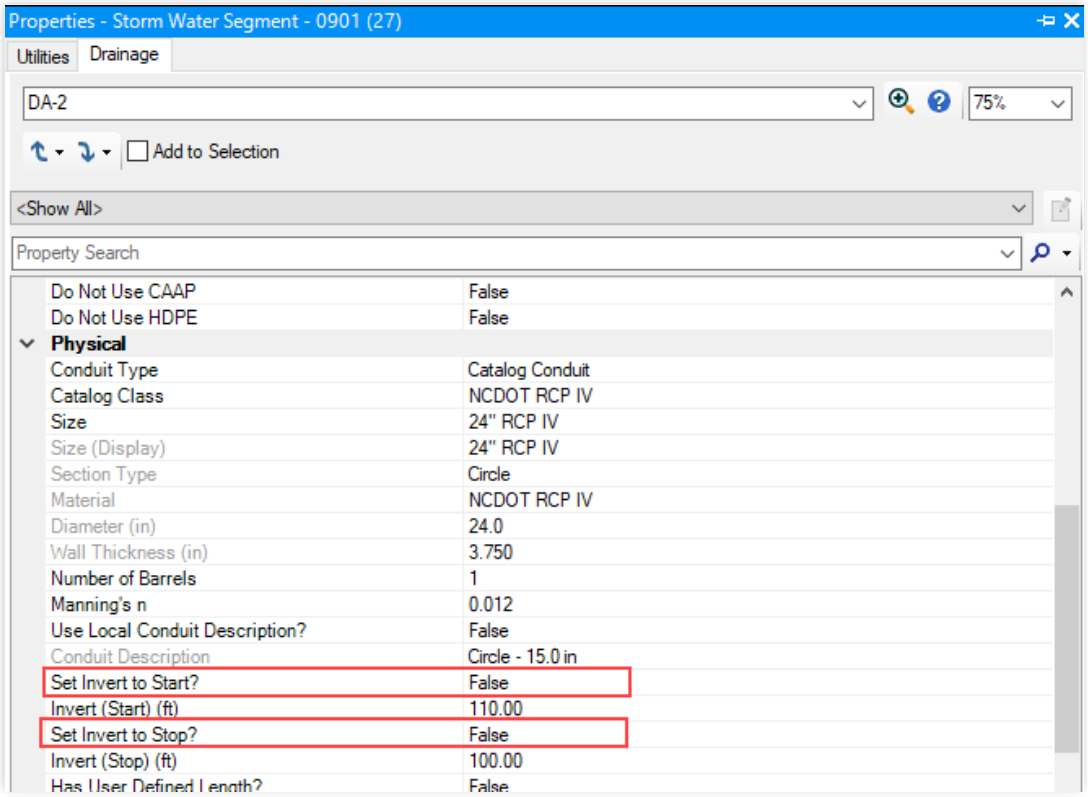
- The headwall or OEP will be placed as an outlet node in the utility properties by default. If the headwall is to be used as an inlet node, place a conduit (**Section 8.1**) and choose the headwall as the upstream node. After placement, the headwall will be changed to an inlet automatically (see screenshots of the utility properties before and after below)
- Before assigning headwall as the upstream node to a conduit:



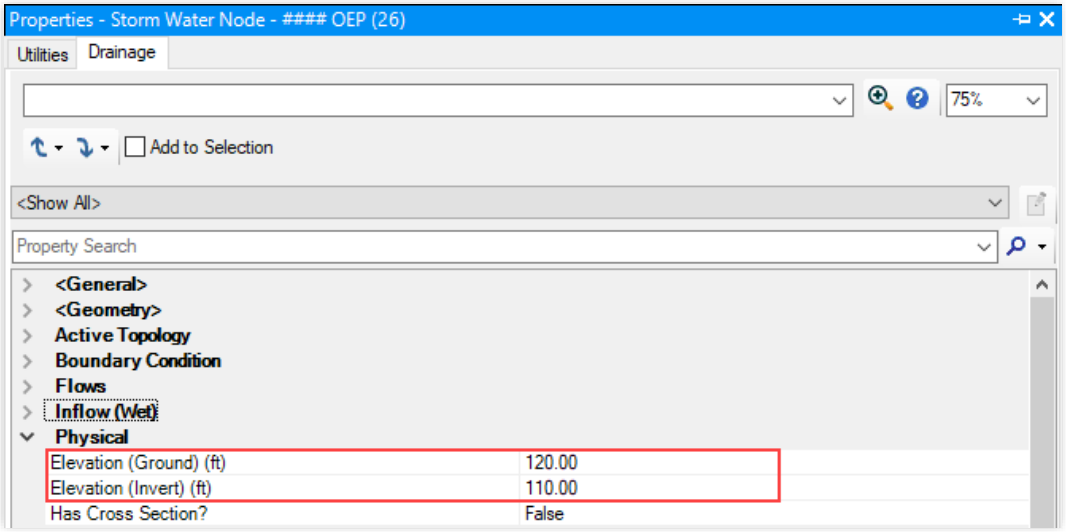
- After:



- Once the conduit is connected to the headwall/endwall/OEP, several other properties will need to be edited as outlined below. **If these steps are not completed the software will incorrectly report the hydraulic grade line elevation at the node.** Note this only needs to be done for headwall/endwall/OEPs.
- Select the conduit and open its utility properties (**Section 3.2**). Navigate to the physical properties "*Set Invert to Start?*" and "*Set Invert to Stop?*" and ensure that one or both are set to false depending on which end the OEP/headwall/endwall is located.



- Select the headwall/open end node and open the utility properties. Navigate to the physical property "*Elevation (Ground) (ft)*"



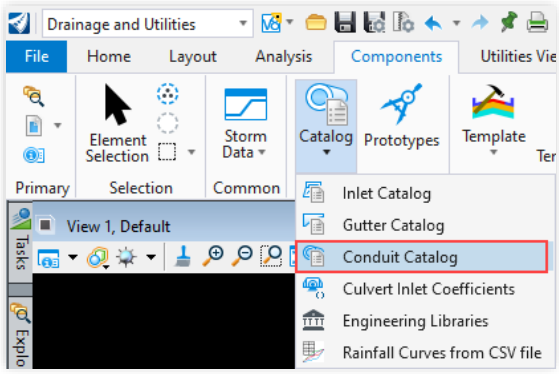
- Change the property "*Elevation (Ground) (ft)*" so that it is much higher than the invert of the pipe (this will not affect 3D view). The reason for this is due to an error where the HGL will only be reported up to the "ground" elevation.
- The designer is welcome to add driveway pipes into the model but it is not required. NCDOT is currently working on a process to have driveway pipes not placed in the model included in the Drainage Summary Sheet.

8.4 Conduits: Background Data

Miscellaneous Pipe Components (For Informational Purposes Only)

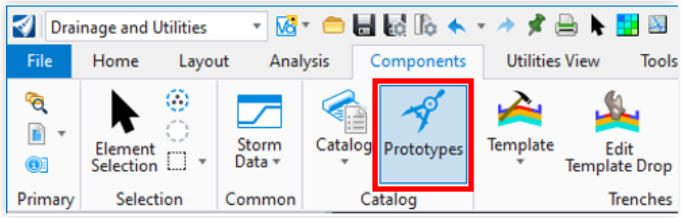
Similar to the node background data and components outlined in **Section 4.5**, conduits have the same two setting locations.

- The Conduit Catalog is where default pipe type and size options are stored. It is located at Components Ribbon Tab > Catalog Drop Down > Conduit Catalog



  - **The catalog should be used for information purposes only. Editing it should be done at the users own risk. It is not recommended to edit the default NCDOT conduits. In rare situations, conduits can be duplicated, and new ones created.**

- Additional Conduit properties and types are also stored in the Prototypes library located at the Components Ribbon Tab > Prototypes



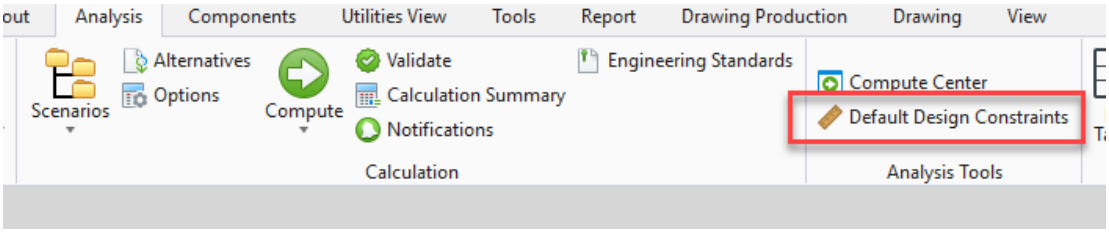
  - **The Prototypes library should be used for information purposes only. Editing it should be done at the users own risk.**

9.1 Pipe Hydraulic Computations: Running Scenarios

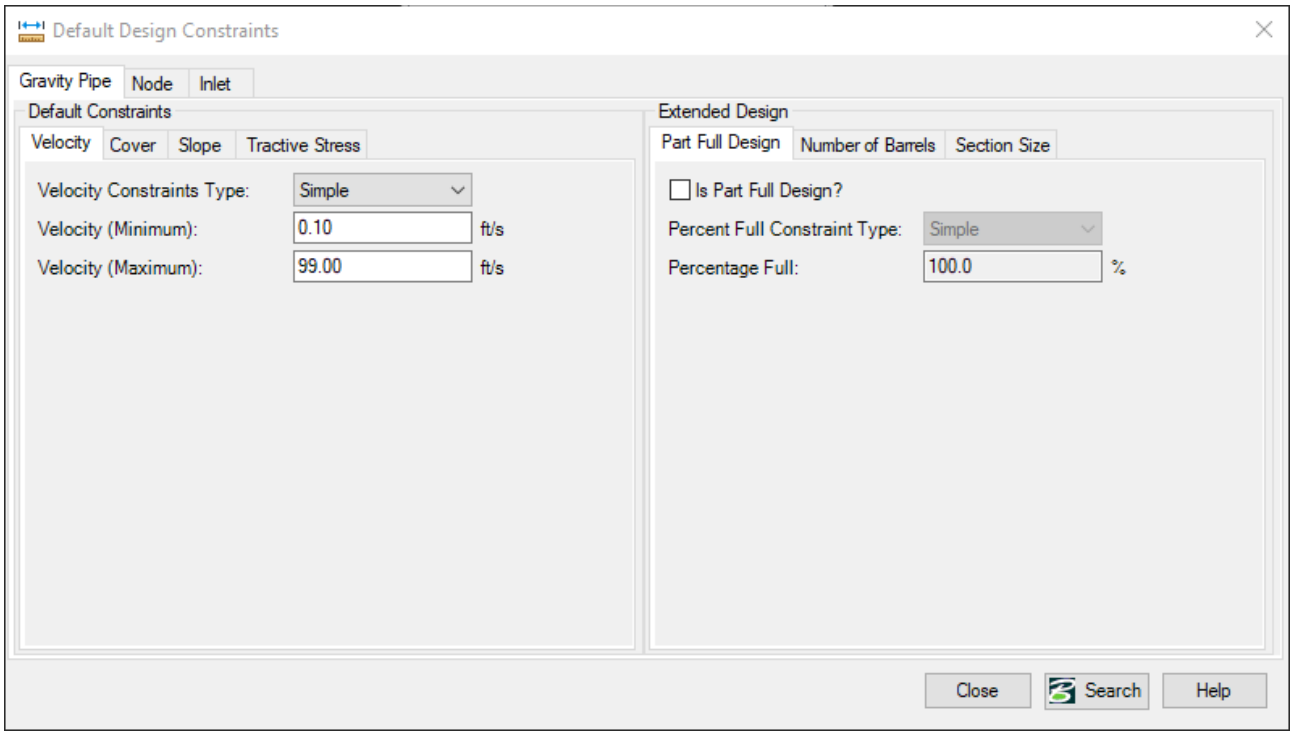
Using the NCDOT Design/Analysis Scenarios to Run 10-yr, 25-yr, 50-yr, etc. Storms

Once all the conduit connections have been placed, a design storm is ready to be applied to the system. ORD Drainage and Utilities can design inverts and pipe sizes automatically or analyze without changing them.

- To generate the proper warnings and design the system to NCDOT / Project specific standards, the default design constraints must be set under the Analysis Ribbon Tab > Default Design Constraints



- The default design constraints dialog box will open. It is recommended these constraints be checked every time a new design .dgn is created.



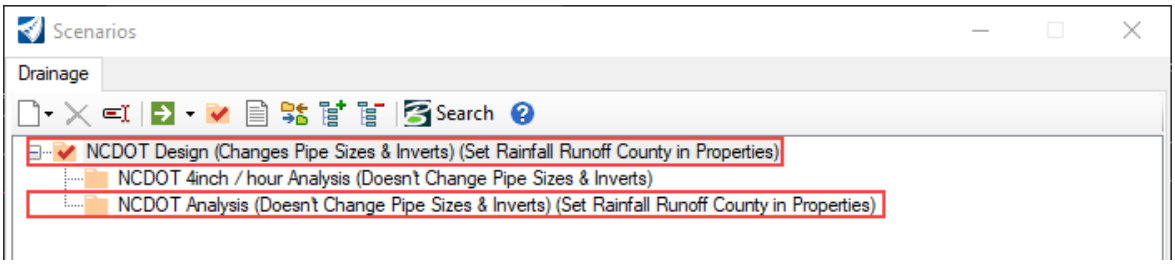
- Below are tables with the preferred NCDOT design constraints

Pipe Design Constraint	Description and NCDOT Recommendation
Velocity Constraints Type	▪ *Can use simple or table format (table allows different pipe sizes to have different velocity constraints)*
Velocity Min and Max	▪ *NCDOT does not have guidance on Min and Max Velocities – These can be left as is. If the user needs to set them, they should be set appropriately for Geographic / topographic conditions.*
Consider Cover Along Pipe Length?	▪ *Leave unchecked - especially when your active terrain model is the existing terrain model*
Measure Cover to:	▪ *Pipe soffit due to NCDOT minimum depth Chart*
Cover Min and Max:	▪ *Set according to most common structure on project (CB, 2GI, Etc) For CB set at 1.75, to ensure 2.0' min. cover under pavement* ▪ *Note: This is the minimum cover set for all pipes. The software will design pipes at this depth measured from the top elevation of the node and generate warnings based on these criteria* ▪ *If a few structures with a shallower minimum depth (such as Type-D 2GIs) are on the project, the software may design them deeper than necessary. The user can then edit the elevations afterwards manually if needed and only a warning message will appear after that*
Slope Min and Max:	▪ *Set min to 0.005 unless in a very flat area and 0.003 is needed. Set max to 0.1 at first to help identify pipes over 10% and then adjust as necessary. Pipe outlets may be above 10% by design and need adjusted manually as well.*
Include Tractive Stress Design?	▪ *Leave unchecked - not needed at this time for NCDOT*
Is Part Full Design?	▪ *Leave unchecked – It is preferrable to design for full flow pipes.*
Allow Multiple Barrels?	▪ *Leave unchecked - multiple barrels are a very specific circumstance*
Limit Section size?	▪ *Checked and set to 72"*

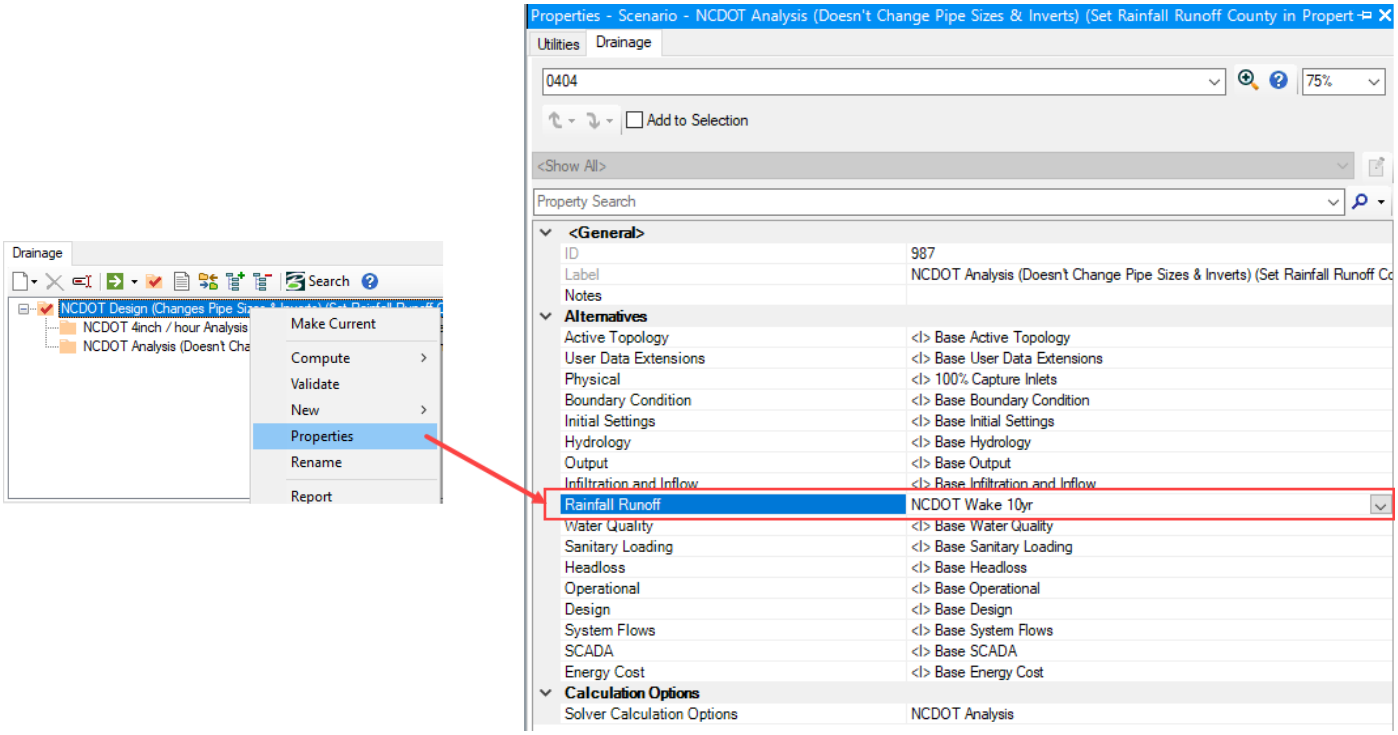
Node Design Constraint	Description and NCDOT Recommendation
Pipe Matching	▪ *Use Crowns - Unless in a very flat, elevation constrained area*
Match line offset	▪ *If needed - but usually set to 0.00*
Minimum standpipe height	▪ *Leave as zero*
Allow Drop Structure?	▪ *Leave Checked*
Use Drop Structure to Minimize Cover?	▪ *Leave checked*
Min Drop Depth	▪ *Set to zero for NCDOT projects*

Inlet Design Constraint	Description and NCDOT Recommendation
Maximum Spread	▪ *Use the spread criteria most prevalent on the project according to the most recent NCDOT Drainage Guidelines.*
Maximum Gutter Depth	▪ *Use 0.5 feet*
Min Efficiency on Grade	▪ *Leave as 1%*

- After the design constraints are set, Open the Scenarios Manager (**Section 7.1**) (Analysis Ribbon Tab > Scenarios > Scenario Manager)

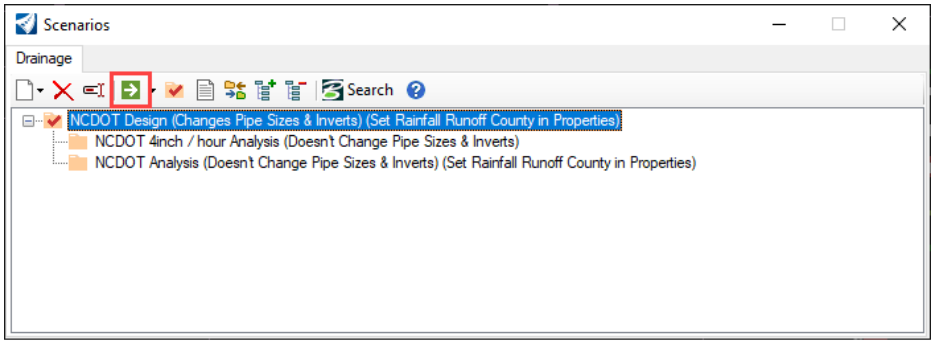


- The two scenarios that will be used for pipe design are outlined in red above and defined below
  - **NCDOT Design:** Run this scenario on a system to have it design and change pipe sizes and inverts using the rules set in the default design constraints
  - **NCDOT Analysis:** Run this scenario to analyze a system without changing pipe sizes or inverts
- Set the rainfall according to the project's location for **both** the design and the analysis scenarios by right clicking them and selecting "properties" as shown below. Start with the 10-yr storm.
- **Do not** change the rainfall runoff alternative for the NCDOT 4 inch/hour scenario.

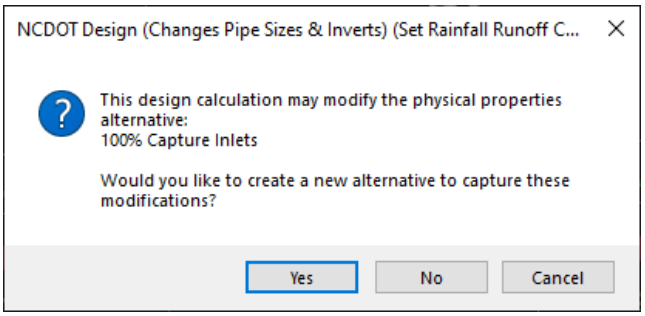


Helpful Hint: Typically the 25-yr, 50-yr and larger return period storms should only be selected for the NCDOT Analysis Scenario and not the NCDOT Design Scenario. The analysis can be run for the 25 or 50-yr storm without changing pipe sizes and inverts and then HGLs can be viewed near sags. If system adjustments are needed near sags or cross pipes that are part of a system they can be hand edited (see **Section 9.3** for hand editing guidance)

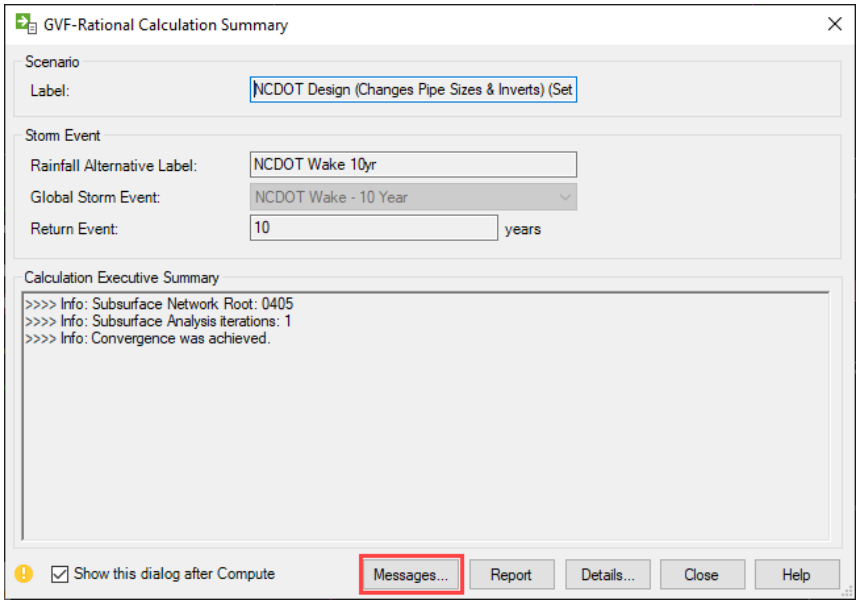
- Select the NCDOT Design or NCDOT Analysis scenario and click the compute button outlined in red below.



- A warning message may appear as shown below if running the design scenario. Click no.



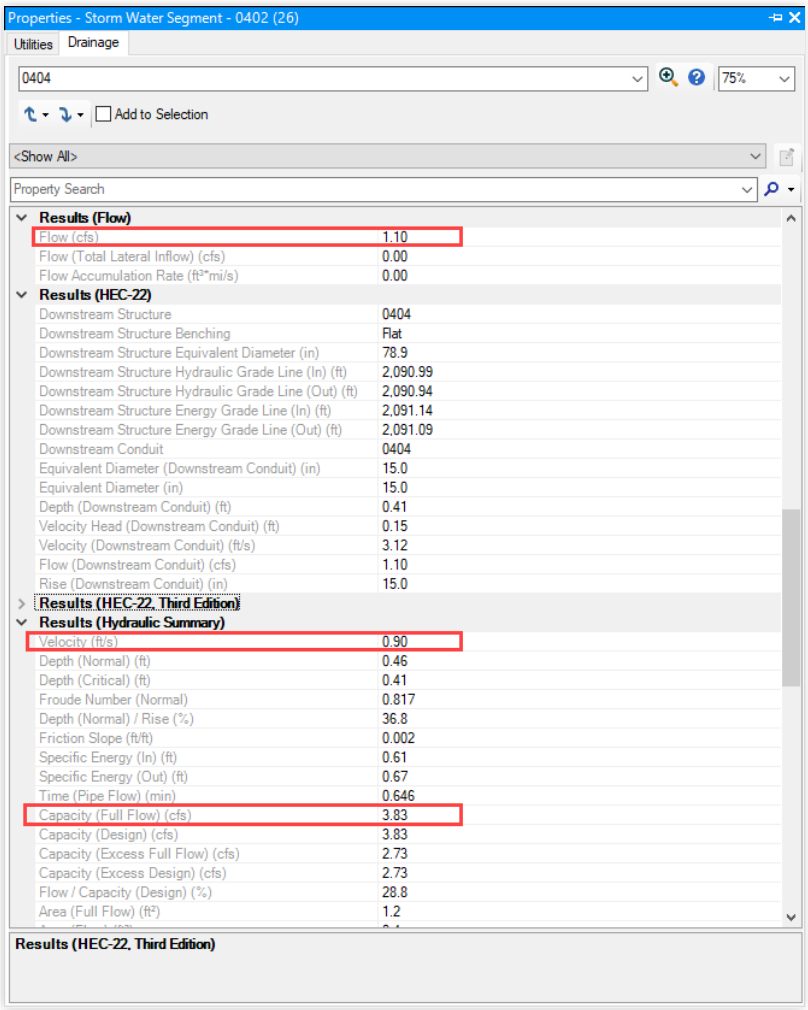
- The calculations will commence and the summary dialog box will open. Click "messages" to view warnings or errors.



9.2 Pipe Hydraulic Computations: Viewing Results

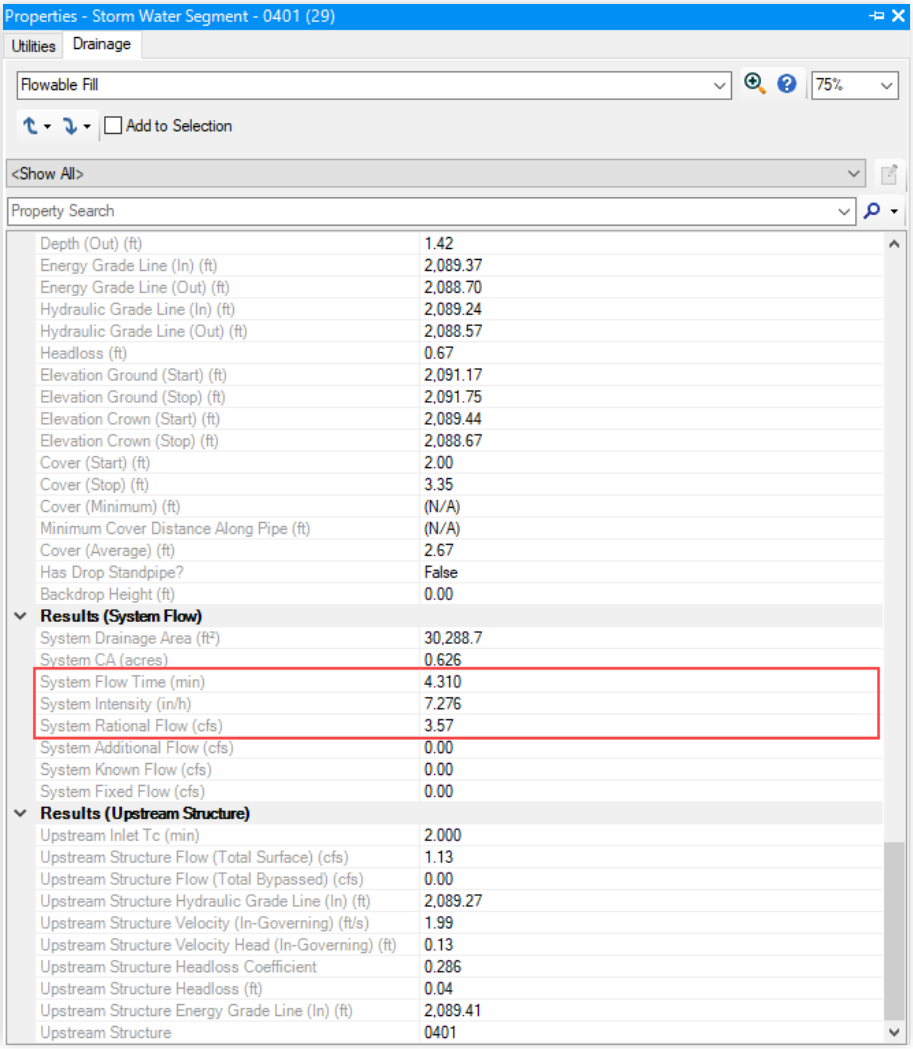
Using the Utility Properties to View Pipe Computations

- After the NCDOT Design / Analysis scenario has been run, typical pipe computations such as pipe flow, capacity, velocity, HGL, headloss and more can be checked by viewing the Utility Properties. Users may also prefer to check these calculations with Flex Tables (guidance on Flex Tables is outlined in **Section 10.1**)
- Open the Utility Properties for a selected conduit (**Section 3.2**) and scroll down to the results sections as shown below.



- Typical properties of interest are shown outlined in red above. Many other calculation variables are available for display.

- **Note:** Earlier in **Section 5.2**, it was mentioned that when a time of concentration (TOC) of less than 10 minutes is entered for an inlet, the minimum TOC (10 minutes) is used for all flow calculations other than the cumulative system flow time. Currently, there exists a display error for the system intensity when the system flow time is less than the minimum TOC (10 minutes). As shown in the screenshot below, the "*System Intensity*" is reported as the intensity associated with the "*System Flow*" time of 4.3 minutes however, the "*System Rational Flow*" is reported as the flow associated with the intensity of the 10-minute time of concentration. The "*System rational flow*" is correct in this situation and the "*System Intensity*" is merely displayed incorrectly and should be ignored.

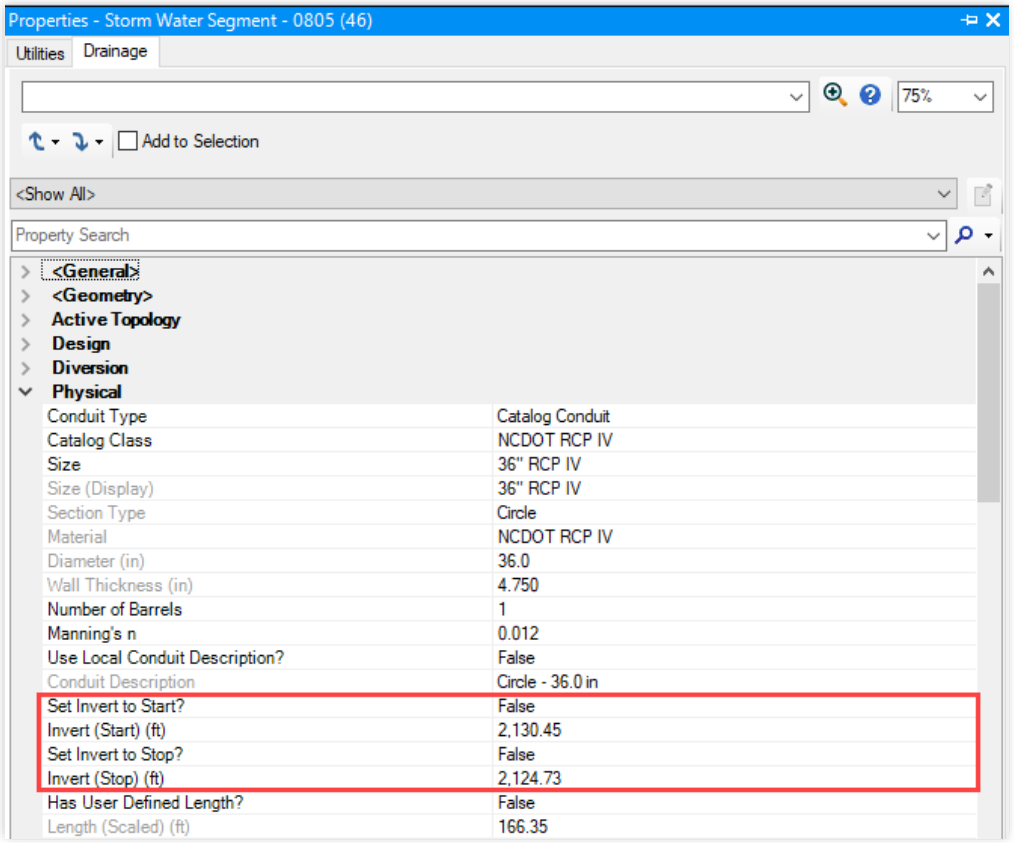


9.3 Pipe Hydraulic Computations: Adjusting the Design

Editing System Inverts, Pipe Sizes and Other Information

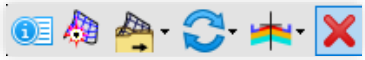
The design computed by the design scenario is not always perfect and will need reviewed and/or tweaked to avoid conflicts, reduce outlet velocities, etc.

- If the user ran the "NCDOT Design" scenario, they can check and edit which inverts and pipe sizes the program changed/designed with either the utility properties (screenshot below) or "Stormdrain_System_All_StormRpt" flex table (**Section 10.1**).

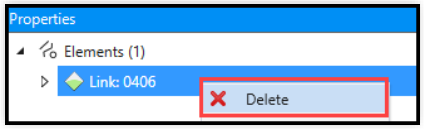


- To change inverts, simply delete the invert value in the "*Invert (start)*" or "*Invert (stop)*" fields and enter the new invert. Note: The "*Set Invert to Start?*" and "*Set Invert to Stop?*" must be set to false if it is not already in order to edit these.
- To change pipe sizes simply use the drop-down list in the "*Size*" field
- Inverts and sizes can also be edited directly within the flex tables (**Section 10.1**).
- Remember that if a pipe material needs to be changed that the recommended process is to delete the conduit and replace it with a new one of that material.

- There are several ways to delete a drainage element or node
    1. Select the element and hover over the element until the quick toolbar for it pops up. Click the Red "X" as shown below.

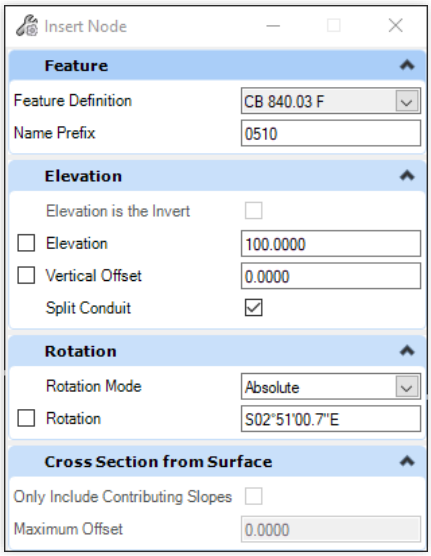


    2. View the element properties. Right click the element > delete (see below)

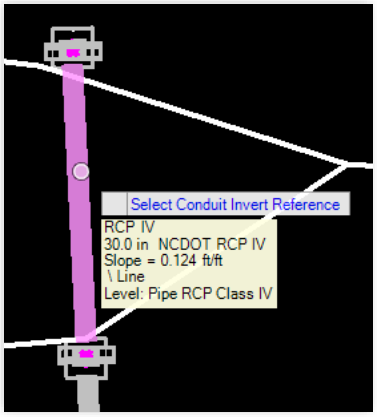


    3. Use the explorer tool (**Section 3.3**) and right click the element > delete

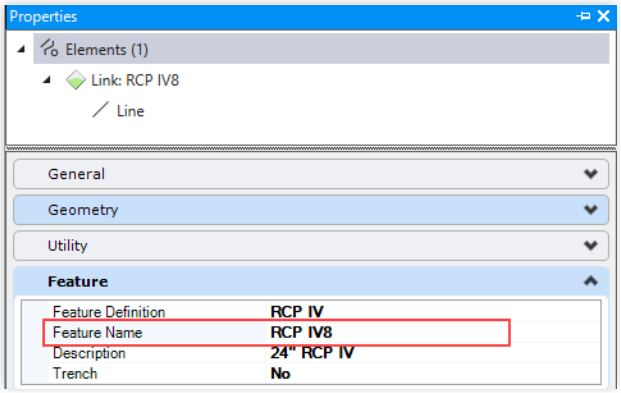
- Another tool that can be useful after the design has been run/completed is the Insert Node tool. This tool can be used to easily insert a node into an existing system and on top of an already designed pipe conduit. The tool will automatically split the conduit in two and interpolate elevations.
- To use this tool, go to the Layout Ribbon Tab > Insert Node. The Insert Node dialog box will open as shown below. Ensure that the "*Split Conduit*" Option box is checked.

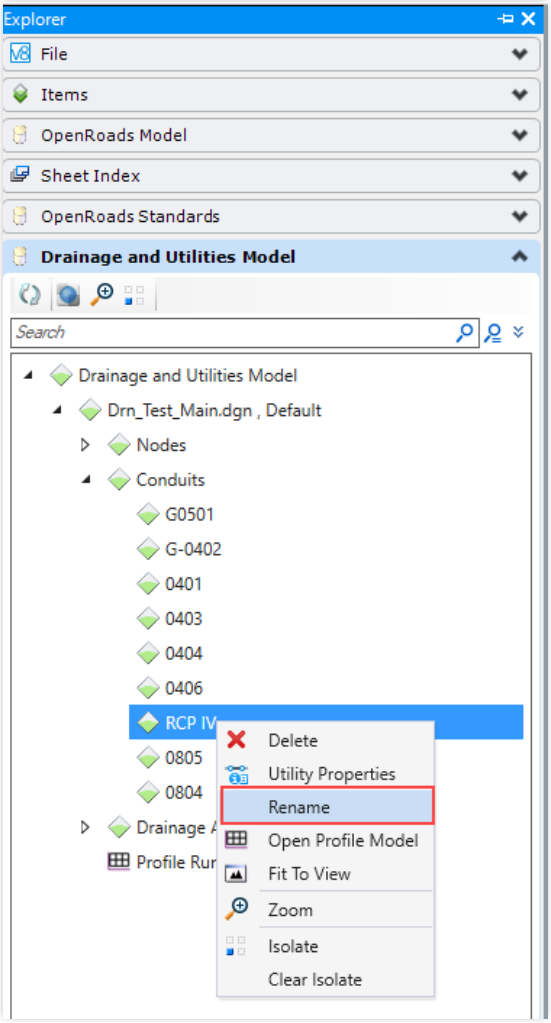


- The process is very similar to the Place Node tool outlined in **Section 4.1** however, when the user gets to the step shown in the screenshot below they will be prompted to select the conduit to insert the node in between



- Select the conduit and then select a location along it to place the node. Once the node is placed the conduit will split in two. The conduit's end points (snap points) will reset and will need to be moved back to the correct sides of the structure by selecting the end point and snapping it to the correct side of the structure.
- Both conduits will be renamed to the default naming convention and will need to be renamed manually. This can be done either in the element's properties as shown below (first screenshot next page) or in the explorer tool (second screenshot next page).





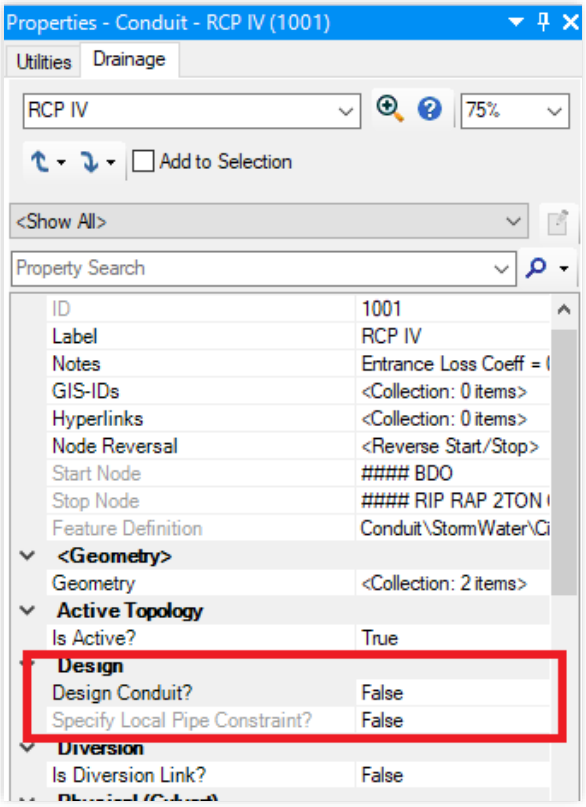
Helpful Hint: Both the property window and the explorer tool can be used to rename any drainage element in the model, not just conduits.

9.4 Pipe Hydraulic Computations: Running Multiple Systems

How to Handle Multiple Systems/Networks and Lock Inverts/Pipe Sizes

Drainage and Utilities lacks the ability to *easily* analyze/design individual drainage networks one at a time when the model contains multiple networks. By default, the design scenario changes inverts and pipe sizes for **every network** in the model and **will overwrite any user modifications to elevations done in Section 9.3** above. To stop this from happening and avoid losing any user modifications, follow the recommended process below.

- Design each system independently. Once a single system is hand modified and designed satisfactorily, the user will need to go to each conduit for that specific system and change "*Design Conduit*" to false. This can be done in the utilities properties window (see below). This must be done before running the design scenario to design other networks or all hand edits will be lost.

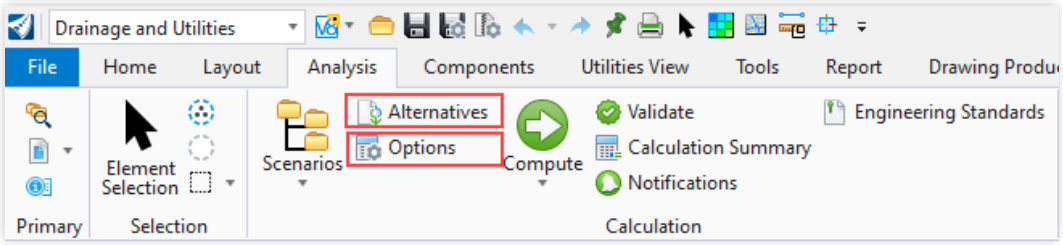


Helpful Hint: For larger systems, custom flex tables can be used to quickly set the "Design Conduit" to false for multiple conduits.

9.5 Pipe Hydraulic Computations: Background Settings

Miscellaneous Hydraulic Analysis Components (For Information Purposes Only)

- Within the Analysis Ribbon Tab and next to the Scenarios icon are two additional components called Options and Alternatives



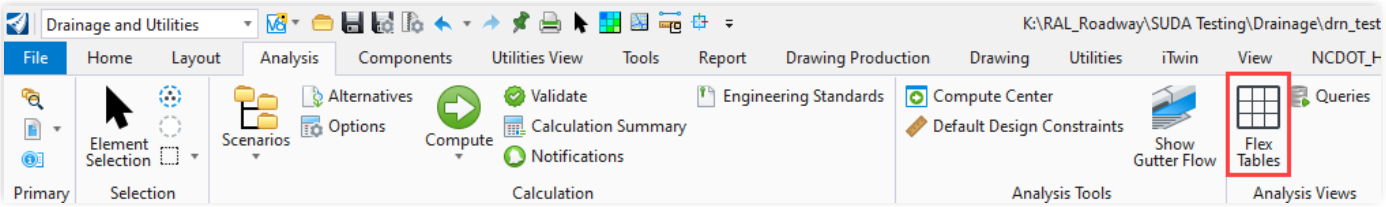
- Options contains what are referred to as "Solvers" which are a key component of a Scenario. They contain more behind the scenes, in depth calculation parameters that will rarely be changed for NCDOT projects.
    - **"Options" should be used for information purposes only. Editing it should be done at the users own risk. It is not recommended to edit the default NCDOT Solvers within "Options".**
- Alternatives contains more behind the scenes variables and ways to create different scenarios. Like options, alternatives will be rarely be changed for NCDOT projects.
    - The most common alternative is the Rainfall Runoff. If the user wished to add a 100-yr, 500-yr storm or custom storm with NOAA rainfall data for a more accurate geographic location, they could create one here.
    - **More advanced or experienced users of Stormcad may be familiar with Alternatives and be able to utilize them to compare different scenarios and designs however, in most cases Alternatives should be used for information purposes only. Editing it should be done at the users own risk. It is not recommended to edit the default NCDOT Alternatives.**

10.1 Flex Tables: Introduction and Creation

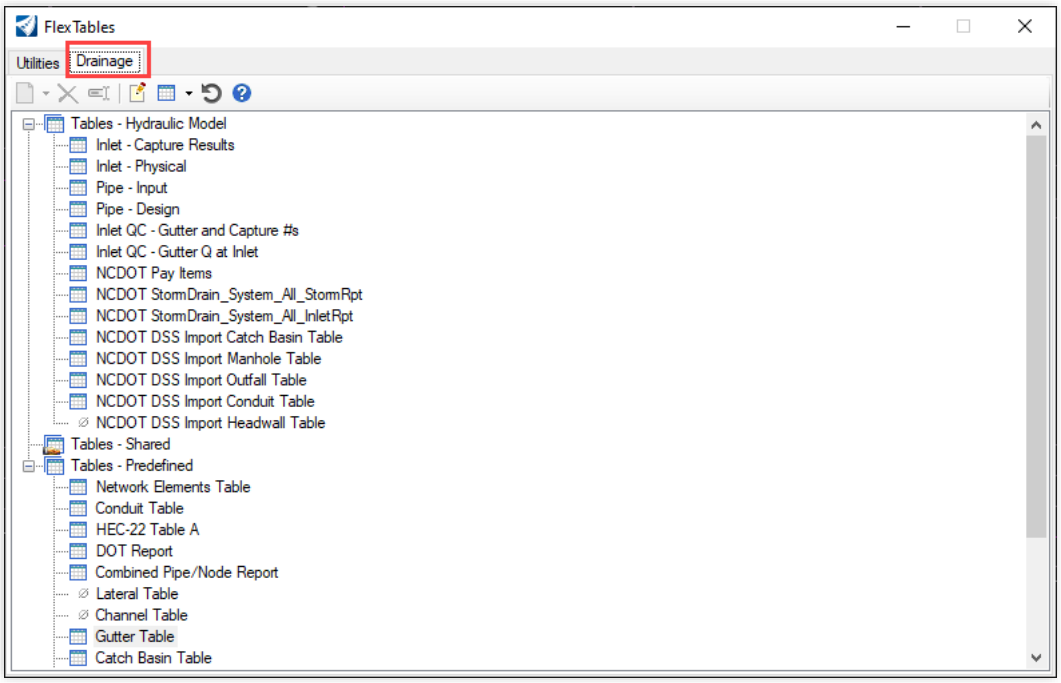
How Flex Tables are Used as a Summary Report

Flex tables are used to generate summary reports on the Drainage and Utility elements. They can also be used to edit properties of drainage elements (elevations, pipe sizes, etc.) in a tabular format.

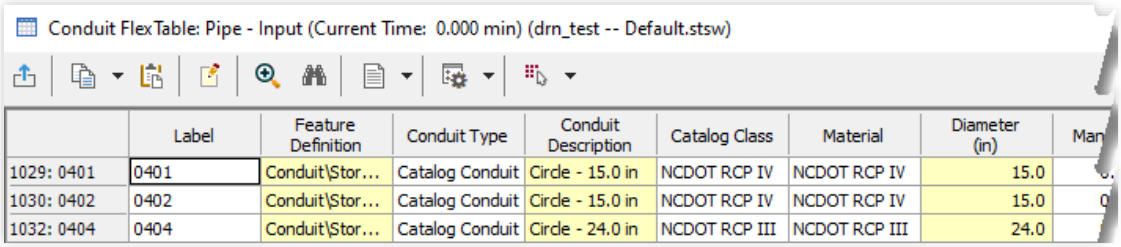
- To access the flex tables, go to the Analysis Ribbon Tab > Flex Tables



- Once the flex table dialog box opens, go to the drainage tab



- Double click on a table to open it. Note the NCDOT standard flex tables available to choose from.



	Label	Feature Definition	Conduit Type	Conduit Description	Catalog Class	Material	Diameter (in)	Man
1029: 0401	0401	Conduit\Stor...	Catalog Conduit	Circle - 15.0 in	NCDOT RCP IV	NCDOT RCP IV	15.0	
1030: 0402	0402	Conduit\Stor...	Catalog Conduit	Circle - 15.0 in	NCDOT RCP IV	NCDOT RCP IV	15.0	
1032: 0404	0404	Conduit\Stor...	Catalog Conduit	Circle - 24.0 in	NCDOT RCP III	NCDOT RCP III	24.0	

- Properties of elements that are not highlighted in yellow can be changed and edited within the flex tables.
- **Note:** The *Storm_Drain_System_All_StormRpt* and other flex tables that provide system time calculations will show the same display error for system intensity as outlined in **Section 9.2** (see screenshot below).

	ID	Label	Start Node	Stop Node	System Drainage Area (ft²)	System CA (acres)	Length (Scaled) (ft)	Upstream Inlet Tc (min)	System Flow Time (min)	System Intensity (in/h)	Flow (cfs)
1051: 0401	1051	0401	0401	0403	30,288.7	0.626	256.98	2.000	4.310	7.276	3.57

10.2 Flex Tables: Standard Calculation Outputs

Printing Flex Tables to Excel and NCDOT Standard Calculation Outputs

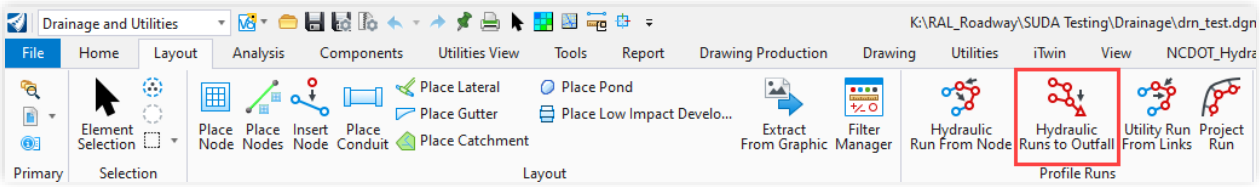
NCDOT is currently in the process of developing the Excel standards for calculations in ORD. Future releases of this document will cover Standard Calculation Outputs here.

11.1 Pipe Profiles: Model Creation

Using Hydraulic Runs to Create Profiles within the Model

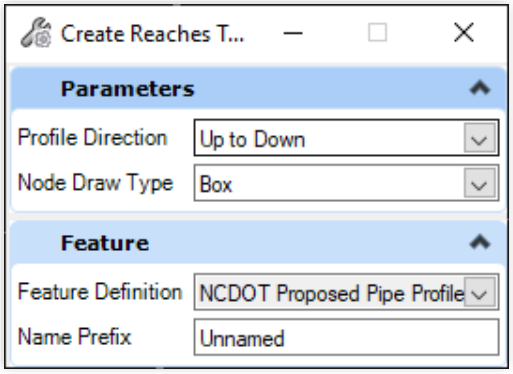
In order to create a pipe profile with accompanying HGL, EGL, proposed grade, existing grade etc., the user must first create what is known as a hydraulic run. A hydraulic run is essentially an alignment that runs along the pipe corridor.

- To create a hydraulic run for an entire system, go to the Layout Ribbon Tab > Hydraulic Runs to Outfall

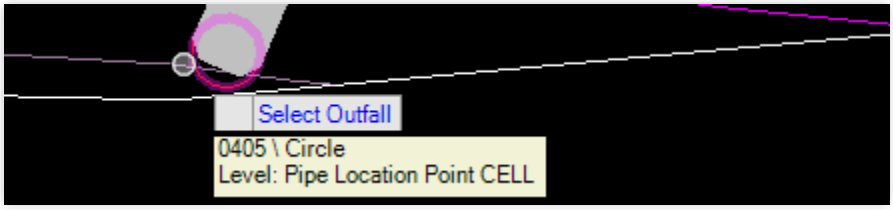


Helpful Hint: "The Hydraulic Run From Node" command to the left can be used to create profiles to/from specific nodes rather than the entire system

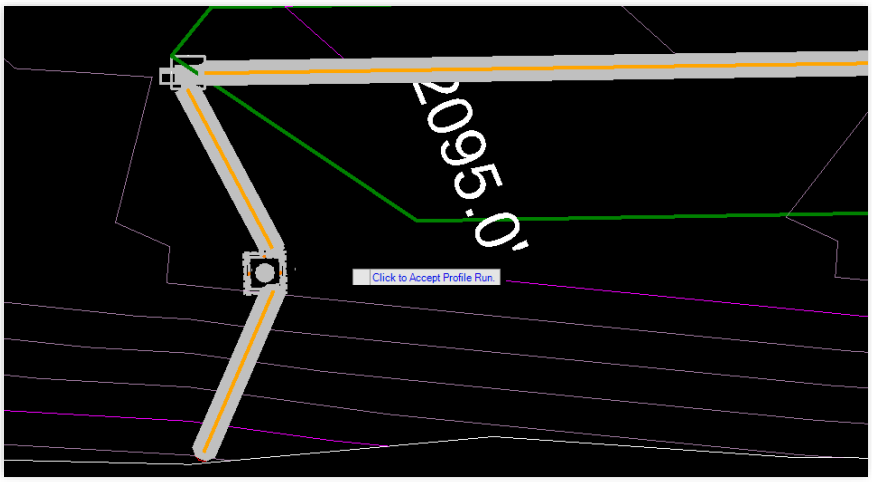
- The Create Reaches to Outfall dialog box will open. The options shown below are the typical inputs. Node draw type "Box" will draw the nodes as boxes in profile view. Ensure to select the feature definition "NCDOT Proposed Pipe Profile."



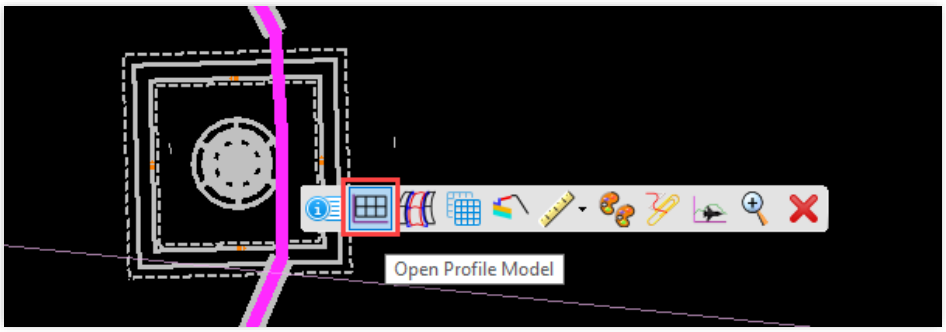
- Select the outfall of the system to create the hydraulic run as shown below



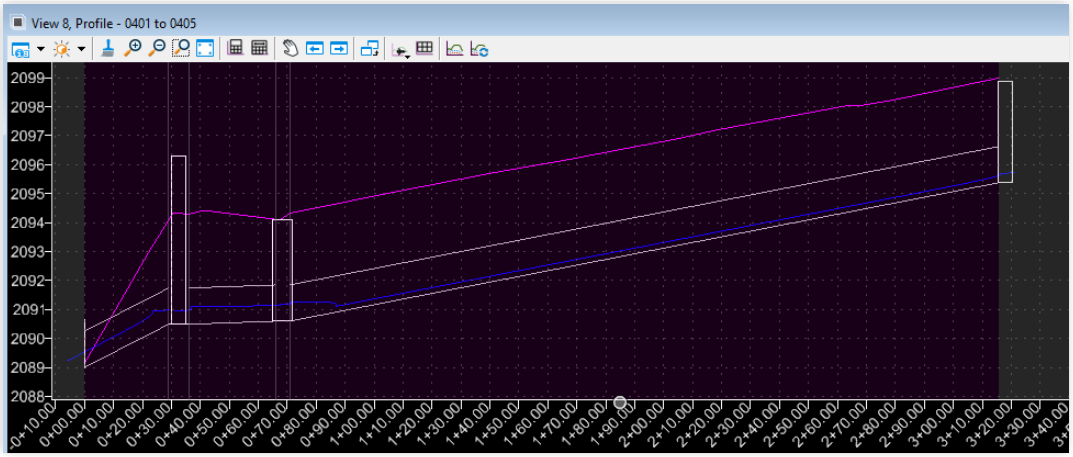
- The proposed hydraulic run will be shown temporarily in orange, left click to accept



- The hydraulic run alignment will be automatically drawn in plan view. It may be drawn under the existing pipes and not visible but should be there. Select it and hover over it until the pop-up toolbar appears. Select "Open Profile Model" as shown below



- Open a separate view and click anywhere within that view to open the profile. The x and y-axis will automatically be generated as shown below. If the design or analysis scenarios have been run the HGL will also be plotted.

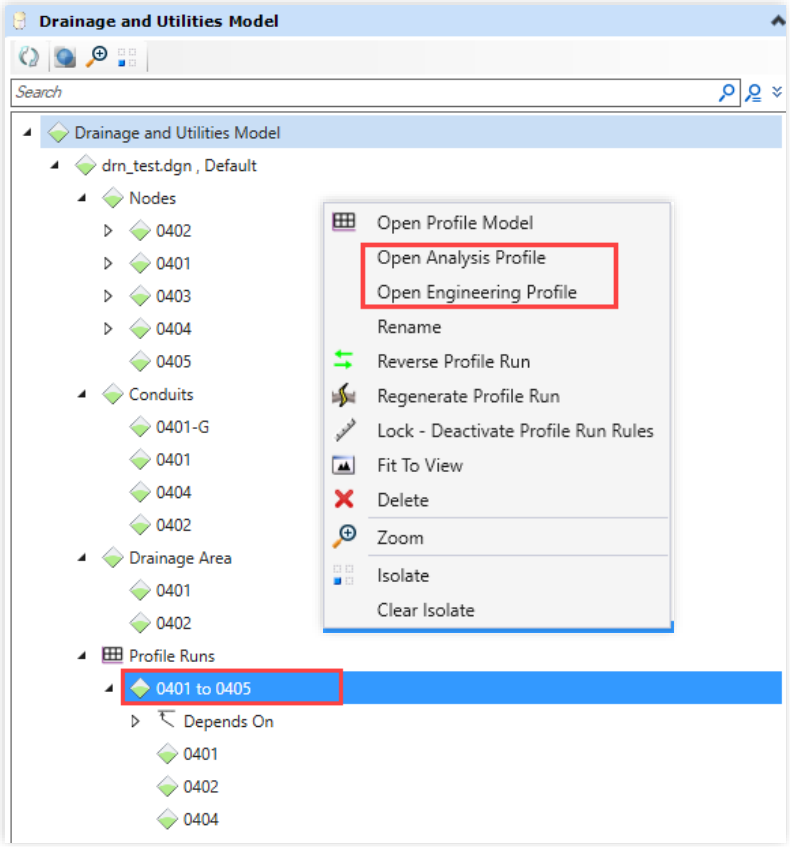


11.2 Pipe Profiles: Engineering and Analysis Profiles

Opening and Customizing the Engineering and Analysis Report Profiles

In addition to the model profile generated above which is only viewable within ORD, there are two other profile types (Engineering and Analysis) which can be generated for paper or export.

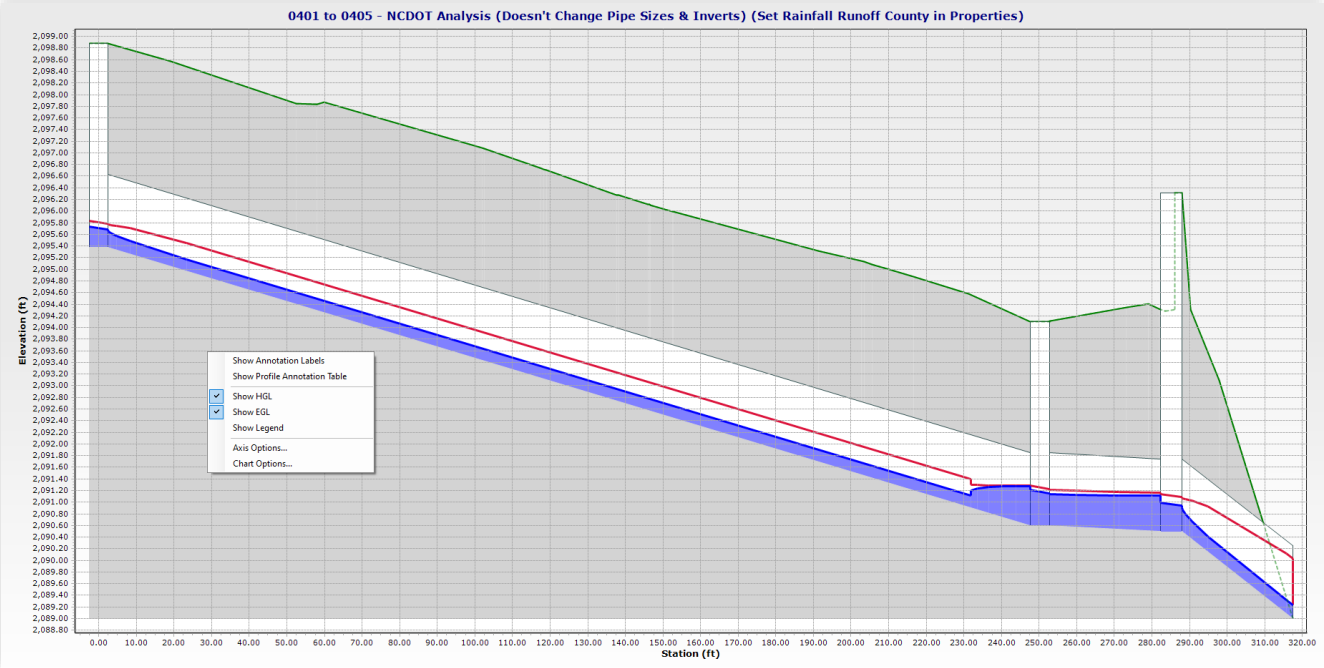
- Open the explorer tool as outlined in **Section 3.3**
- Expand the Drainage and Utilities Model section within explorer and all of its subcomponents. All the nodes, conduits and catchments contained within the model can be viewed here as well as the newly created hydraulic profile run.



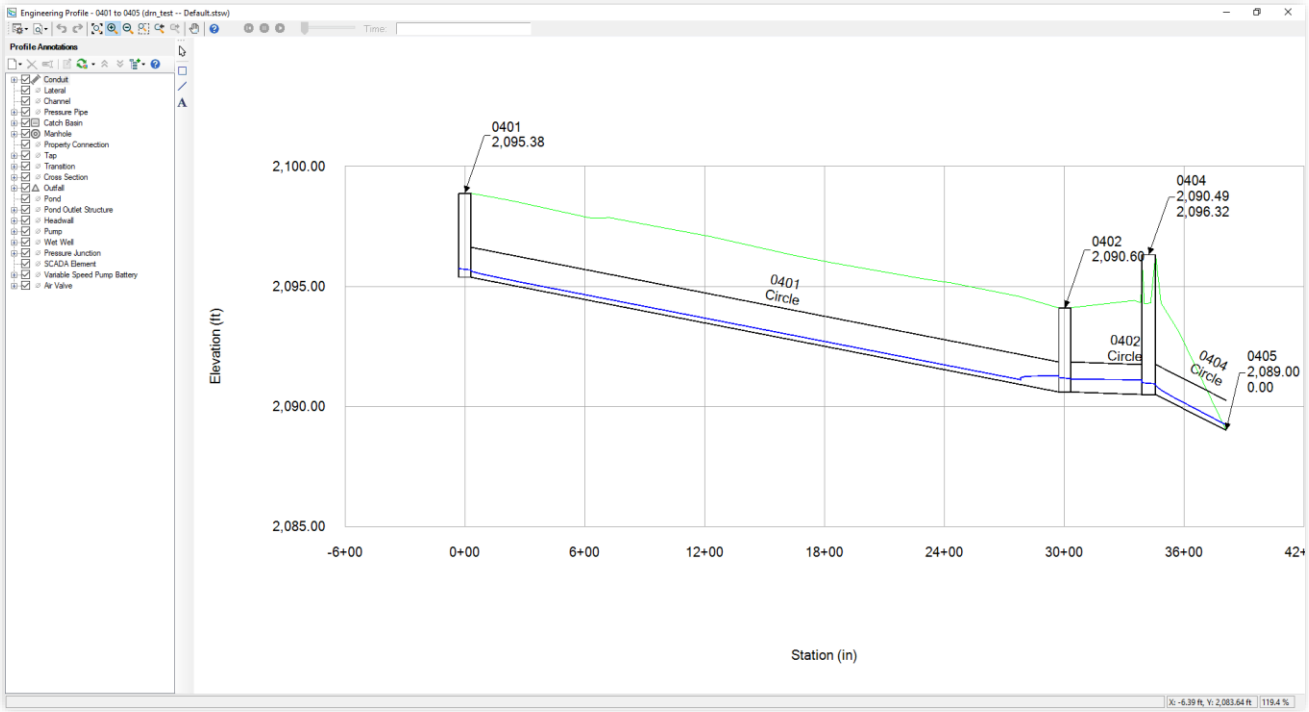
- Right click on the profile run (example outlined in red above) and the three profile options will be shown at the top.

*(The first option, "Open Profile Model" is what was opened in the previous **Section 11.1**)*

Analysis Profile



Engineering Profile



The user can customize the labels and data shown on each to their liking and then print to a .pdf or export it.

12.1 Labeling: Drainage Labels for Plan Sheets

Standard Labeling for Sheet Views and Drawing Scales

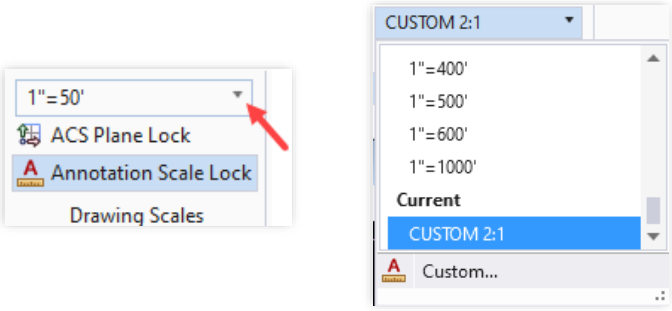
NCDOT is currently in the process of developing standards for notes, callouts, and labeling in ORD. Future releases of this document will cover drainage labeling here.

12.2 Labeling: Analytic View

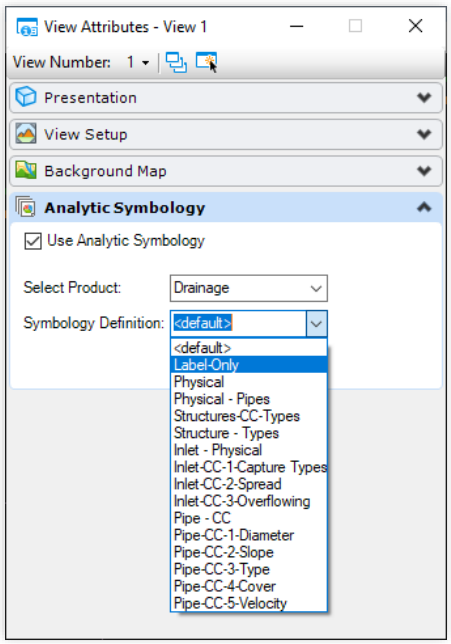
Turning on Temporary Analytic Labels and Gutters for Drainage Components

The following steps below outline how to turn on temporary labels within the model space to view basic drainage element names and other properties without having to open the properties tool for each one.

- First, make sure to change annotation scale to something close to 1:1 or smaller to view the analytic view labels properly. **Note: Changing the annotation scale to 1:1 or smaller should only be done temporarily and will make other labels too small to be viewed. The standard scales for all other labels to be viewed and placed properly is are the typical 1"=20', 1"=40' 1"=50', etc.**
- Under the Drainage and Utilities Workflow go to the Drawing Production Ribbon Tab > Drawing Scales tool group. Change the scale as depicted in the screenshots below.

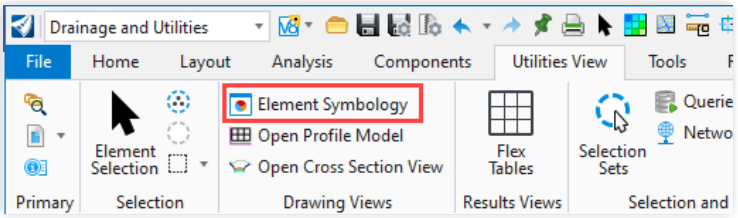


- To turn on and off Analytic View, press CRTL+B or go to the View Ribbon Tab > View Attributes.

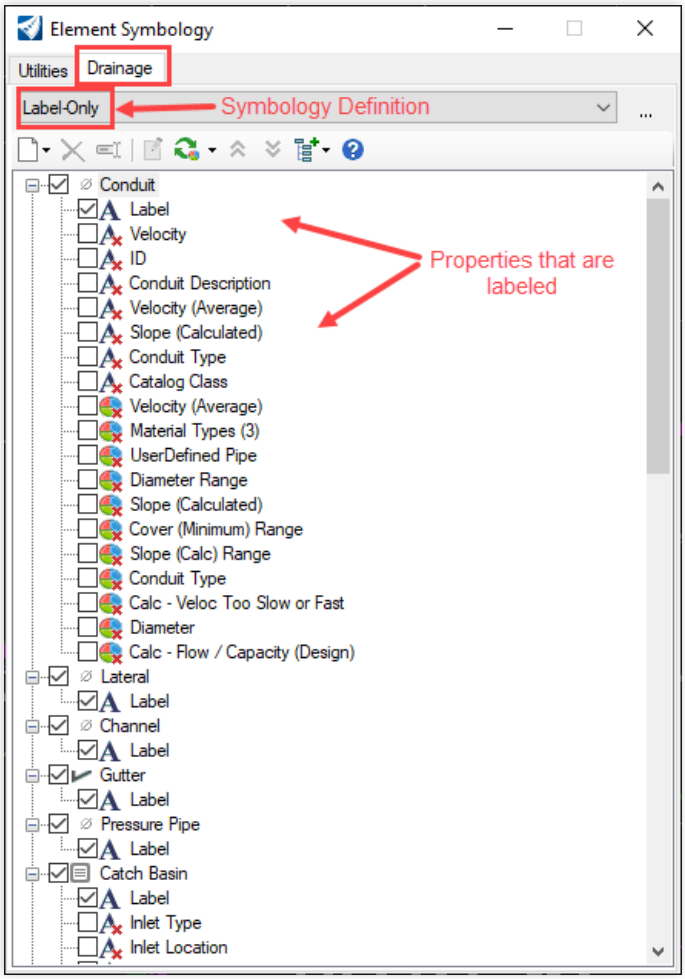


- Ensure that "Select Product" is set to "Drainage"

- The "Symbology Definition" controls what set of drainage items are labeled and what type of properties (node name, catchment name, top elevation etc.) are shown in the labels.
- To edit and create Symbology Definitions use the tool located in the Utilities View Ribbon Tab > Element Symbology.



- The Element Symbology dialog box will open as shown below. Symbology Definitions with different property labeling preferences can be edited or created here.

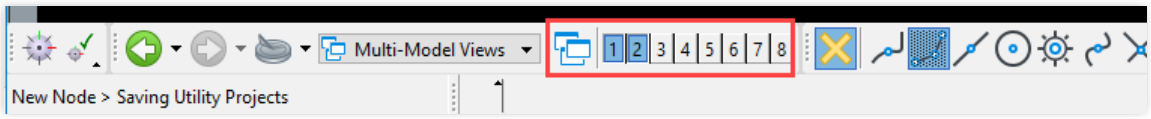


13.1 Views: Opening a 3D View

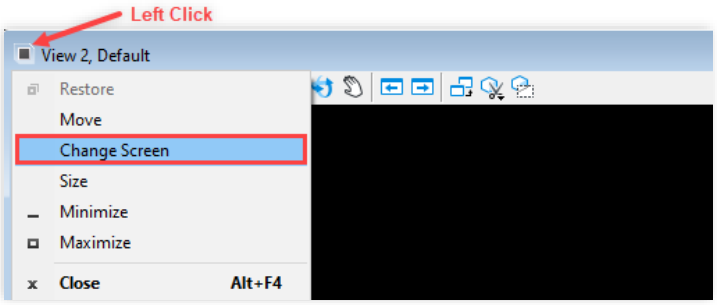
Using a Second Screen or View to Check a Drainage Model in 3D

Every Drainage and Utilities model contains 2D and 3D elements/linework. In earlier sections, it was recommended that the 3D reference of the active drawing (and 3D references of roadway design files) be turned off because design was being done in the 2D plan view. It can be helpful, especially with two screens, to have a 3D view open in tandem with the 2D view. The steps below outline how to open a 3D view.

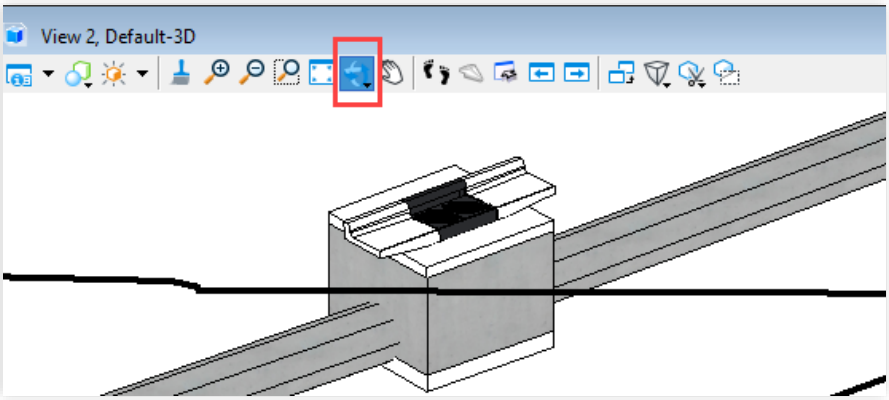
- Open a second view using the view toggles toolbar (typically docked on bottom or top of the screen)



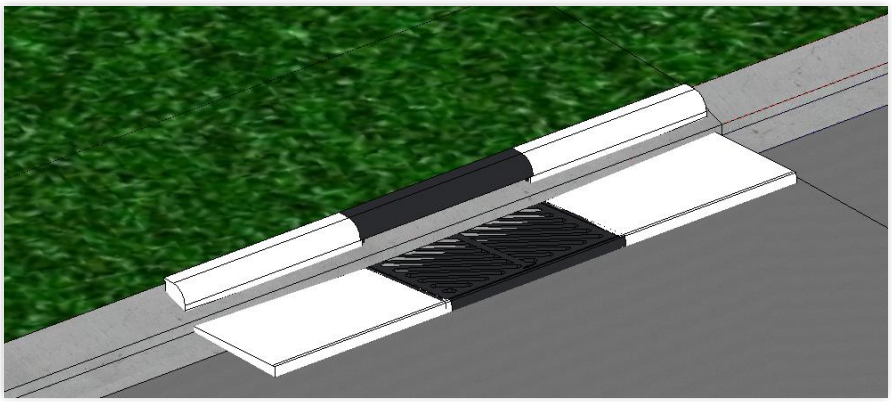
- The view will open. To change screens, left click the top left corner as shown below



- Right click anywhere in the view and select View Control > 2 Views Plan/3D. The 3D view will open in view 2.
- Use the view rotation tool as shown below to rotate around in 3D.



- When active in a 3D view, notice that the some of the attached 2D versions of references do not appear. The user will have to click in the view they want to be active in to turn on and off references for that view.
- Reference displays are independent of each other in a 2D vs. 3D view. For example, a 3D reference can be turned on in the 3D view but will remain turned off in the 2D view and vice versa.
- The 3D view can be helpful to spot check box depths or any major elevation errors in reference to the roadway corridor model (see screenshot below showing a catch basin relative to the roadway gutter in 3D).



- If utilities are put into a 3D utility model the 3D view can be a powerful tool in identifying and visualizing utility conflicts with the storm drainage.

14.1 Proposed Ditches: Standard Ditch Modeling Process

Designing, Drafting, and Coordinating with Roadway to Model Proposed Ditches

NCDOT is currently in the process of developing standards for ditch modeling and design in ORD. Future releases of this document will cover ditches here.

15.1 Quantities: Pipe Removal, Plugs and Flowable Fill

Standard Process for Drawing in and Quantifying Removal, Flowable Fill and More

NCDOT is currently in the process of developing a process for quantifying and reporting pipe removal, plugs, and flowable fill in ORD. Future releases of this document will cover those topics here.

15.2 Quantities: Drainage Summary Sheet

Exporting All Drainage Quantities and Creating the DSS

NCDOT is currently in the process of developing a process for generating a drainage summary sheet in ORD. Future releases of this document will cover this topic here.

16.1 References and Contacts

This document is meant to be a living document that will be updated as Bentley makes improvements and NCDOT projects are fully transitioned to ORD software. User feedback, questions, and bugs/error reporting will be an important aspect of determining the best practices and improvements that can be made. Please use the contacts and links below as additional resources.

Document Updates

NCDOT is currently in the process of developing an email list that will send out notifications for when this document is updated.

Contacts

NCDOT is currently in the process of developing a webpage for ORD Drainage and Utilities which will have contacts, supplemental forms, and FAQs

Additional Guidance / References

NCDOT

- Hydraulics Unit Homepage
- CADD Services ORD Homepage
- Instructional Videos (link currently in development)

Bentley

- Bentley OpenRoads Designer CONNECT Edition Document
- Bentley Subsurface Utilities CONNECT Edition Help
- Bentley OpenRoads YouTube Channel

Comments and Questions

Please direct comments and questions regarding this guide to HydraulicCADDSupport@ncdot.gov and a project team member will reach out.

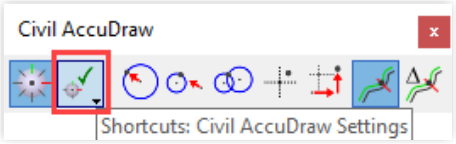
Appendix A – Customized Civil Accudraw

Creating A Custom Civil Accudraw Option for "Unlinked" Station and Offset

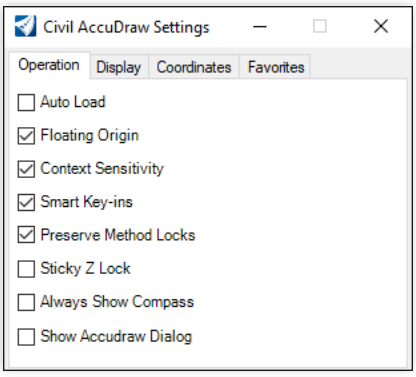
In Section 4.1, it was outlined that nodes are typically placed with a station referenced to an alignment and the offset is referenced to the same alignment. There are certain scenarios where it may be useful to have the node's station and offset referenced to separate alignments. For example, a node's station can be referenced to the -L- alignment but the offset referenced to the curb alignment. This can be useful when the -L- alignment is straight and the curb is curved or similar scenarios.

To have this option, the user will have to create a custom option in the Civil Accudraw settings as outlined below.

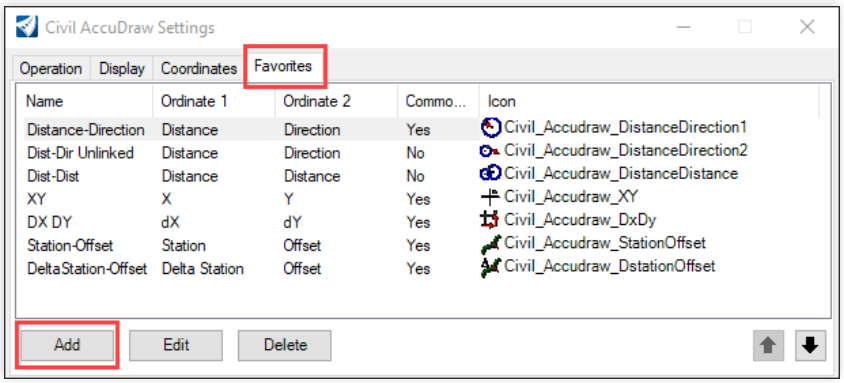
- Open the Civil Accudraw toolbar as outlined in Section 3.4 and select the Accudraw settings as shown below.



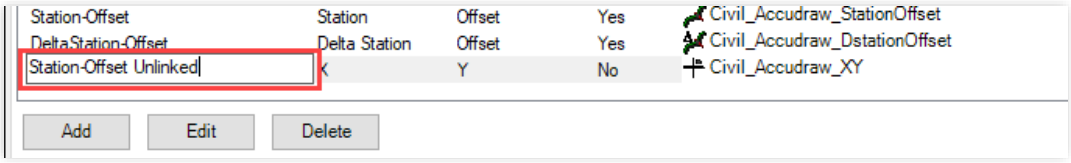
- The Accurdraw settings will open as shown below.



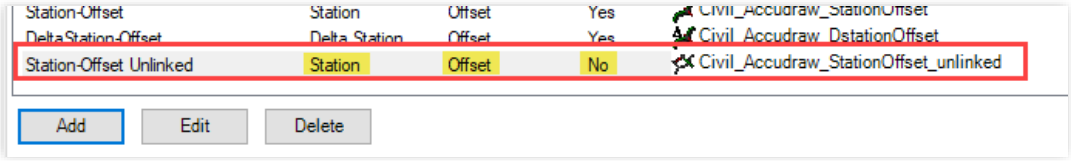
- Select the favorites tab as shown below and click "add".



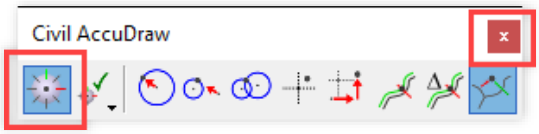
- Name the new favorite "Station-Offset Unlinked" as shown below.



- Input the settings for the newly created "Station-Offset Unlinked" as shown highlighted below.



- If the new option does not show up, toggle on and off Civil Accudraw or exit and open the toolbar again as shown boxed in red below



Helpful Hint: Ensure Civil Accudraw is associated with the current alignment first by using a command that will display it such as "draw line." If it is not associated with the current alignment, press the "O" key ("O" stands for "origin") in either the station or offset data field, hit enter, and it will prompt the user to select an alignment. With this newly created option in particular, this process is more finicky, and one error is that sometimes entering "O" in the station field may not do anything. If this happens, follow the steps below:

1. <Reset> (right click)
2. Toggle on the regular **station-offset** option
3. Enter the "O" in the offset field and not the station field
4. Select the alignment the station should be linked to
5. Toggle on the **station-offset unlink** option
6. The station should now be linked to the alignment you chose in step 4

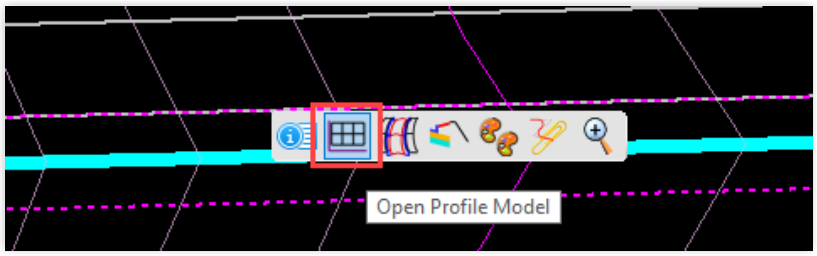
If these steps fail, some form of alternating between the **station-offset** option and **station-offset unlink** option and entering "O" in the offset field may correct the issue.

Appendix B – Roadway Profile Reports

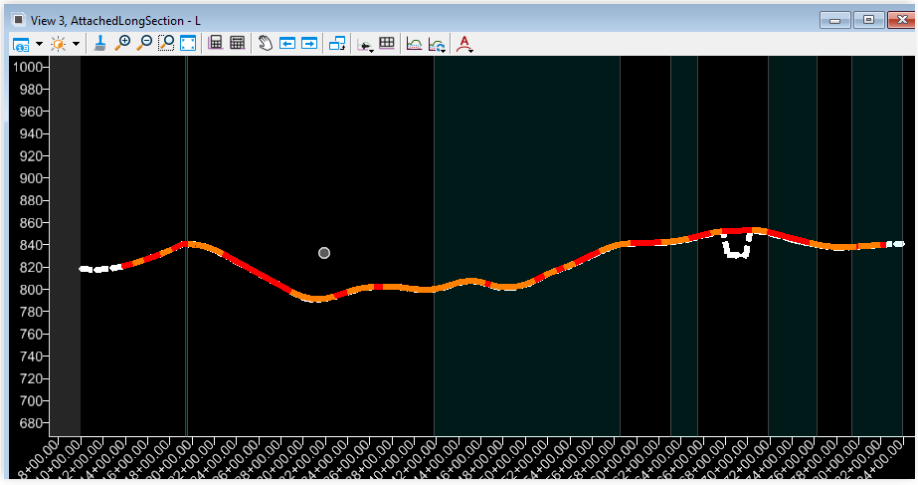
Using Roadway Profile Reports to Locate Crest / Sag Locations

It is often necessary for the hydraulic engineer to mark sag/crest locations on the plan and use these locations to place nodes. To view a vertical geometry report of a profile, follow the steps below.

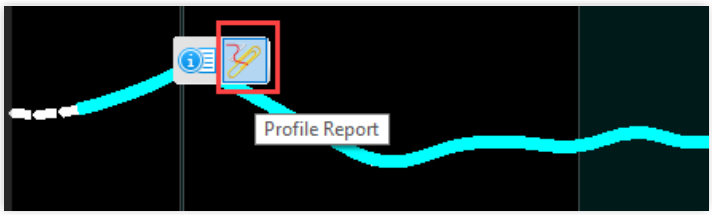
- Ensure the roadway alignment is referenced into the current drawing and turned on for the main view.
- Open a second view as outlined in **Section 13.1**
- Select the alignment and hover over it until the toolbar appears as shown in the screenshot below.



- Select the "Open Profile Model" as shown outlined in red above.
- OpenRoads will prompt the user to select a view. Select the second view screen by clicking anywhere in it. The profile will open.



- Select the profile and hover over it until the toolbar appears next to the cursor as shown in the screenshot below.



- Open the profile report. Select the "VerticalAlignmnetReview.xls" report tab (on the left). The crest and sag locations will be printed out on with the code "**VHP**" (Vertical High Point) and "**VLP**" (Vertical Low Point) with the station and elevation printed to the right of them as shown in the screenshot below. Other vertical geometry data is also printed out such as VPT, VPI, etc.